

MultiCam® Router User Manual

Introduction	IN 1 – 9
Fast Starts	FS 1 – 38
Keypad and Functions	KF 1 – 41
Menu Items	MN 1 – 119
Options	OP 1 – 26
Connections	CN 1 – 19
Software	SW 1 – 32



Read this manual before using this product.

Failure to follow the instructions and safety precautions in this manual
can result in serious injury or death.

Keep this manual in a safe location.



Innovation. Quality. Performance. Best in Industry. Worldwide.

Original Instructions

(a) The business name and full address of the manufacturer and of the authorized representative is as follows:

MultiCam Inc
1025 W Royal Ln
DFW Airport TX 75261

Erik Ketele
Ketele NV
Mechelsesteenweg, 1-3
2540 Hove
BELGIUM

(b) The machinery is designated as Router 1000/3000/5000/7000/8000 Series.

(c) The EC declaration of conformity is attached.

(d) The above CNC machine is designed to modify raw materials that can be used in manufacturing.

(e) The maintenance and repair information is attached.

(f) The workstation likely to be occupied by the operator is a warehouse-like area with high ceilings and heavy flooring and approximately 36" (914.4mm) between the edges of the machine and any auxiliary units appropriate in a warehouse-like area.

(g) The machinery will be used to modify raw materials that can be used in manufacturing.

(h) **WARNING:** The CNC machine should not be used as a table or desk, heater, ladder, washing station, or prank item as the machine has the ability to crush body parts and materials and can cause serious bodily damage up to and including possible death.

(i) The assembly, installation, and connection information is attached.

(j) To reduce noise and vibration, the operator may install machine or vibration isolation pads.

(k) The technician performing the installation will follow an installation checklist as well as an instruction checklist for training operators. This information is attached.

(l) The residual risks inherent in the CNC machine are operator-specific and can include cuts, scratches, burns, amputations, and death resulting from non-adherence to the warning labels placed on the machine and operator-specific association and experience working with CNC machinery.

(m) The protective measures to be adopted by the user include clothing that is not loose or long; minimal to no jewelry and no long-hanging accessories or wardrobe-specific materials; protective eye wear and ear wear; long hair restrained; gloves or comparable hand-covering when handling sharp or hot parts; avoidance of alcohol, drugs, or any other substance that may decrease judgment or alertness; adherence to warning labels placed on the machine as well as operator-specific association and experience working with CNC machinery; avoidance of



Innovation. Quality. Performance. Best in Industry. Worldwide.

placing hands on the tabletop during the cutting sequence; and observance of proper lock-out procedures when completing any maintenance or other work on the machine.

(n) The essential characteristics of the tools are sharp points and sides, blunt edges, dense material, and heavy components.

(o) The conditions that allow the machine to meet the requirement of safety during use, transportation, assembly, and dismantling when out of service, testing, or foreseeable breakdowns are the solid base on which the machine is built, the covers over moving parts, the packaging for shipment, the possible requirement of a crane or forklift to move such pieces as the gantry, the lock-out procedure if followed correctly, optional safety mat and light curtain devices that will halt the motion of the machine, and the automatic stop of all forward motion in the event of a fault.

(p) The instructions for ensuring safe transport, handling, and storage operations include the customer hiring a rigger to facilitate moving the machine; the use of a forklift or crane to move the machine from the delivery van or truck to the final location; a complete inspection of packaged parts to be completed by the installation technician and the customer; information regarding any shipping damages reported immediately to the carrier; insertion of leveling feet so that the machine is at the lowest level possible; and secure anchoring of the machine components to the floor with adequate space for ventilation.

(q) The operating method to be followed if the machine is involved in an accident or breakdown includes the operator or installation technician securing the work area so that the installation technician may determine the cause of the malfunction and troubleshoot a possible workable solution. If a blockage is likely to occur, then the operator or installation technician will halt any forward motion until it is determined that the blockage can be prevented.

(r) The adjustment and maintenance information, as well as the preventive maintenance measures to be observed are as follows:

Safety glasses and hearing protection **should be worn at all times** while operating the machine, and long hair should be **restrained with a cap or net** when near the machines. Operators should never leave the machine unattended during the cutting sequence, and labels should be observed at all times.

All machines should be kept clean when not in use. The build-up of chips and dust on the system should be cleared daily with special attention focused on the gear rack and bearing rails.

All of the bearings should be greased at least twice a month based on an 8-hour work day. The bearings should be greased more often if the machine is in use more than 8 hours a day.

The Lead Screw Z-axis should be lubricated with a non-aerosol, silicon-based lubricant. Operators are cautioned **against** using the lithium grease provided by **MultiCam** when lubricating Lead Screws as this may lead to premature wear of the Lead Screw. Ball Screws, however, can be lubricated with lithium grease.

The filters on the electronics enclosure and vacuum pump should also be cleaned.



Innovation. Quality. Performance. Best in Industry. Worldwide.

(s) The instruction designed to enable safely completed adjustment and maintenance, as well as the protective measures to be taken, is as follows:

The **MultiCam** CNC machine will provide years of productive service if it is maintained properly. There are daily, weekly, monthly, quarterly, and yearly maintenance steps required for each machine based on a 40-hour work week. **MultiCam** recommends that any system used in excess of 40 hours a week should have the maintenance schedule adjusted accordingly.

Daily

The **MultiCam** CNC machine should be cleaned off each day, and the water lines should be inspected. All chips should be blown or vacuumed off the machine. Close attention should be paid to the rack area since chip build-up in the rack can cause stalling problems. Customers who operate systems with a Lead Screw Z-axis should lubricate the screw with a non-aerosol, silicon-based lubricant. Operators should **not** use WD-40 on the Z-axis screw for lubrication. Using anything other than what is recommended will result in the premature wear of the Z-axis screw. With Ball Screw systems, oil or lithium grease is an acceptable lubricant.

Weekly

The **MultiCam** CNC machine should be cleaned thoroughly each week. The filter on the control box should be vacuumed. Any build-up in the racks or rails should be blown out. The filter or filters on the vacuum hold-down systems should be removed and cleaned. All oil levels (if applicable) in the vacuum hold-down system should be checked and filled if needed. On MG Series routing systems, the Y-axis Ball Screw should also be lubricated with oil or lithium grease. MT Series routing systems will need to follow this procedure for the X-axis Ball Screw as well.

Some older routing systems use a Lead Screw on the X- and Y-axes. These systems should be lubricated with a non-aerosol, silicon-based lubricant. Using anything other than what is recommended may lead to premature wear of the Lead Screw.

Bi-Monthly

Operators should grease all the linear bearing cars at least twice a month for the **MultiCam** CNC machine. Jogging the machine while applying slight pressure in the opposite direction will help force the grease into the bearing cars and incorporate the grease throughout all of the bearings.

Monthly

The racks, Y- and Z-axes screws, and X-axis screws on the MT Series routing system should be cleaned with a scrub brush and degreaser once a month following the same lubrication procedure described earlier in addition to the daily and weekly cleaning for all **MultiCam** CNC machines. Once the racks are clean, operators should apply a bead of the lithium-based grease.

Quarterly

The X- and Y-axes covers should be removed from the **MultiCam** CNC machine every three months so that operators can inspect the transmission or gearboxes for wear. Operators can engage the pinion all the way into



Innovation. Quality. Performance. Best in Industry. Worldwide.

the rack by loosening the mounting bolts of the transmission or gearbox and pushing the assembly up. The tabletop bolts should be checked to ensure they are all tight. Filters in the electrical cabinet should be cleaned.

Bi-Annually

The electrical cabinet should be checked during the bi-annual maintenance performed on the **MultiCam** CNC machine. Operators should turn off the power before opening the cabinet and then vacuum out any debris that may have entered the cabinet during regular working conditions. Operators should not use compressed air when cleaning out the cabinet as this action may cause unwanted particles to short out some of the electrical components.

MultiCam recommends that the belts on all transmissions and Lead Screw Z-nuts be replaced every other year.

- (t) There are no specifications of spare parts to be used that may affect the health and safety of operators.
- (u) The information on airborne noise emissions is attached.
- (v) The machinery does not emit non-ionizing radiation that may cause harm to persons, in particular persons with active or non-active implantable medical devices.



Innovation. Quality. Performance. Best in Industry. Worldwide.

Installation Checklist

These machine installation procedures should be followed in order by a **MultiCam** Technician, and the CNC machine should be fully functional at the end of the installation. Three asterisks (***) indicate optional steps that may not be applicable to all CNC machines.

Receiving Preparation

1. Machine unpacked & inventoried (suggest customer take digital pictures)	6. Ethernet communication cable run
2. Shipping damage claimed & replacement parts ordered if any	7. Air pressure to machine must be checked. Incoming Air Pressure _____
3. Machine & fume extraction units in final location w/proper space requirements	8. Fume extraction provided ***
4. Leveling feet on	9. Computer provided
5. Proper electrical to disconnect switch from disconnect box on machine, fume extraction unit, & auxiliary units	10. Process gases, consumables, & materials for training with & 3/8 airline hoses

Mechanical Installation: PHASE I

1. Level table (as low as possible).	5. Re-adjust belts (if necessary).
2. Remove all axis covers.	6. Install the pendant to the control box.
3. Reseat the X-axis motor assemblies in the rack.	7. Slightly loosen the bolts connecting the X castings to the gantry.
4. Connect the motor connectors for XA, XB, Y, and Z to the control box.	

Electrical Installation

Make sure the voltage setting is close to 40Vdc without going over.

1. Measure the incoming voltage at C/B with wires 100/L1 and 102/L3.	6. Check the AC voltage at pins 23 & 24 for approximately 26-28Vac for MD4 and approximately 53-55Vac for HP4. If the level is too high, power down to adjust the transformer.
2. Adjust the transformer with the power OFF. Move wire 112 to matching incoming voltage at locations 210, 220, 230, or 240.	7. Power down and attach. Connector N back to the controller box.
3. Check for loose/disconnected wires (including gantry head and spindle box) with the power OFF.	8. Power up the machine.
4. Disconnect the Connector N from the top of the silver controller box inside the cabinet with the power OFF.	9. Adjust the transformer and try again if the voltage is too high and an error message shows on the keypad.
5. Turn the circuit breaker ON and OFF.	10. Check the voltage at the keypad by pressing Shift , Help , and Six . Refer to the Schematics and adjust the transformer if necessary.



Innovation. Quality. Performance. Best in Industry. Worldwide.

Voltage Chart

Incoming at Switch
(L1,L2,L3) _____

Transformed Power
(Pins 23&24) _____

Mechanical Installation: PHASE II

Electricity is required.

Slightly loosen gantry bolts – Loosen both sides of the X-axis (moving gantry only).

Hard Home – If machine will not find limits, then see Help Docs.

Verify Resolutions – See Help Docs for more information on how to check resolution

Verify Table Parameters – Jog the full distance in X-, Y-, & Z-axes. Make sure that the gantry & head do not make physical contact with the end stops.

Verify the cutting head is parallel to the table.

Square Table – Use a pen taped to the side of the head to make 4 points to verify the square of the gantry. The bigger the perfect rectangle or square, the more accurate.

Tighten Gantry Bolts – Tighten both sides of the X-axis (moving gantry only).

Suite4 Machine Software Installation

	Reboot Machine – Check for red letter errors.
	Capture NVRAM – Check the factory Flash settings before making any modifications. Save to Computer and Tech computers.
	Check Init and Firmware Version – Type v & press <i>Enter</i> . Capture the screen or write down the information. Make updates if necessary.

Interface Board	_____	Init File Version Loader	_____
Controller Board Revision	_____	Init File Version STD	_____
Firmware Version (MCIF)	_____	MD4 Version	_____

	Help Files – Manuals & Help Docs		JobNameServer – Confirm operations.
	JobReporter – Confirm operations.		JobEditor – Confirm operations.
	ConnectionManager – Confirm operations.		JobConsole – Check if Auto Preview works & if Job Monitor is on.

ConnectionManager Setup

	1. Open ConnectionManager.		6. Highlight the XMI file for the machine and select <i>Open</i> .
	2. Right-click on No_Connection_Info next to the machine to be added and select <i>Add</i> .		7. Select <i>Yes</i> at the notice.
	3. Highlight either an existing connection if the new machine is being added to the network or <i>None</i> if this is a new setup.		8. Refer to the ConnectionManager dialog and locate the specific machine connection.
	4. Check the box next to Auto Launch JobNameServer.***		9. Right-click on the connection and select <i>Report a Problem</i> .
	5. Check the box next to Set Preferred Neti using the Current Neti if the customer wants this computer to be in charge of all communications.***		10. Set up the following email settings: • <u>Email</u>: email address of the contact person for the machine • <u>Server</u>: mail.multicam.com • <u>To</u>: suite4@multicam.com

Mechanical Installation: PHASE III

Option Installation

	Fume Extraction – Adjust the air flow.		Bar Code
--	---	--	-----------------

Run a test file to verify machine operation.



Innovation. Quality. Performance. Best in Industry. Worldwide.

Introduction

[top](#)

Customer Responsibilities for Safety	2
Power Lockout Instructions	4
Labels	5

Customer Responsibilities for Safety

As the user of the **MultiCam** routing system, each customer **must** follow all safety procedures that are clearly outlined when operating each machine and its parts. Failure to do so may result in damage to property or personnel and could even result in death. These machines are powerful and should always be treated with care.

Customers should make plans for safeguarding themselves and their work areas at the point of operation as all **MultiCam** machines have been designed and constructed to operate under specific parameters relating to the particular application. As a result, ANSI B-11 *Safety Standards* states that “It shall be the responsibility of the end-user (buyer) to provide, and ensure the use of a guard, guarding device, awareness barrier, awareness device or shield...” in order to maintain the required level of protection.

MultiCam has available certain safety shields and guards applicable to our machines. Please contact our office for styles, types, and prices.



Personal Safety Precautions

All customers should become familiar with all aspects of their machines.

Any person who operates or does any maintenance on this equipment should be aware that safety procedures are an important part of the daily job. Customers and related personnel should learn how the equipment functions and be able to respect the capabilities of the machinery. Anyone working on or around the equipment should understand the potential losses associated with mishandling the machinery and its parts and be able and willing to follow all safety precautions. Sudden movement, loud noises, horseplay, etc. must be avoided, as such distractions may result in unsafe conditions.

Accidents can occur if clothing or other articles become entangled in the cutter or other moving parts of the machines. The following suggestions, if followed, will reduce the chance of having these types of accidents:

1. Wear **approved eye and hearing protection** at all times when operating the routing system.
2. Restrain long hair with a **cap or net** when near the machines.
3. Avoid wearing neckties and scarves during machine operation.
4. Avoid wearing loose fitting clothing during machine operation.
5. Avoid wearing hanging jewelry during machine operation.
6. Wear gloves only when handling sharp or hot parts.
7. Avoid operating this and any other equipment if affected by alcohol, drugs, or any other substance or condition that may decrease judgment or alertness.
8. **Observe and follow all safety signs** on the machine and in the surrounding areas.
9. Avoid placing hands on the tabletop when the spindle is turned on.
10. [Lock out the incoming power supply](#) when any type of maintenance or other work is being performed on the machine.

Work Area Safety

Fire extinguishers should always be readily accessible, and operators should always familiarize themselves with the fire prevention recommendations for each component of the CNC system. It is important to always keep the work area **clean and uncluttered**. Oil, debris, or water on the floor can cause unsafe conditions. Customers should be sure that all work areas are free of hazardous obstructions and that all tools and other equipment are returned to their proper storage place when not in use. Operators should never leave the machine unattended during the cutting sequence.

Cutter Safety

Cutting tools are very sharp and should be handled accordingly. Machine operators should inspect tools before each use and discard any defective tools. Failure to properly handle tooling can result in serious injury.

Installation Safety

An electrician must read and understand the electrical schematics prior to connecting the machine to the local power system. Connecting the wrong voltage power will void the warranty. All switches should be turned to the OFF position before power is connected. The main disconnect switch should always be locked in the OFF position if the machine is left unattended. When the machine is installed, the electrician or customer should be sure that all motors rotate in the correct direction.

[Introduction](#)

Power Lockout Instructions

MultiCam machines are equipped with a built-in main power lockout device. If any kind of repair work or maintenance is being performed to the machine or control cabinet, the operator should **disconnect power** from the machine **before starting work**.

1. Turn the main disconnect switch to the OFF position.
2. Flip out the plastic lockout latch.
3. Insert any padlock into one of the holes of the lockout latch. With the plastic lockout latch in this position, the switch cannot be moved to the ON position.



1



2



3

4. Throw the bulkhead switch to the OFF position.
5. Open the control cabinet.
6. Make sure the LEDs on the drives and inverters are not lit. It may take a few minutes for the light from the LEDs to fade away.
7. Make sure the incoming voltage for L1, L2, and L3 is 0v.
8. Follow the operator's standard Tag Out procedure in tagging the disconnect switch.

[Introduction](#)

Labels

Each machine produced by **MultiCam** includes certain cautions, warnings, dangers, and notices placed on the moving parts of the machine (e.g., gantry, carriage) as well as the stationary parts (e.g., electrical enclosure). These labels are in place to encourage employees to observe safe operating practices at all times. The text on these labels is almost always capitalized and presented in bold format. Any auxiliary equipment (e.g., vacuum pump, chiller) will include vendor-specific labels advising the customer of proper safety practices regarding those units as well. The serial number label is unique to the machine. The most common labels are identified below.

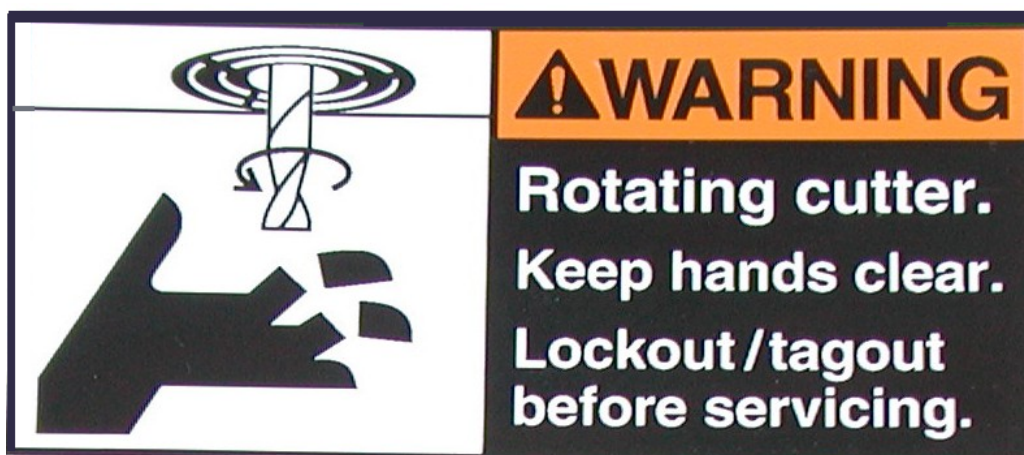
Cautions

Caution labels indicate the potential for minor or moderate injury and are usually yellow with black writing. These labels are normally placed on the gantry, axis covers, and any other covers that are located at the front of the machine.

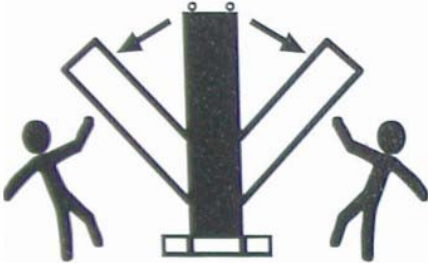


Warnings

Warning labels indicate the potential for death or serious injury and usually show the word WARNING in black writing against an orange background. These labels are normally located in areas where serious injury could be sustained, such as the carriage.



	! WARNING Crush hazard. To avoid tip over, this equipment must be securely fastened to the floor.
---	---

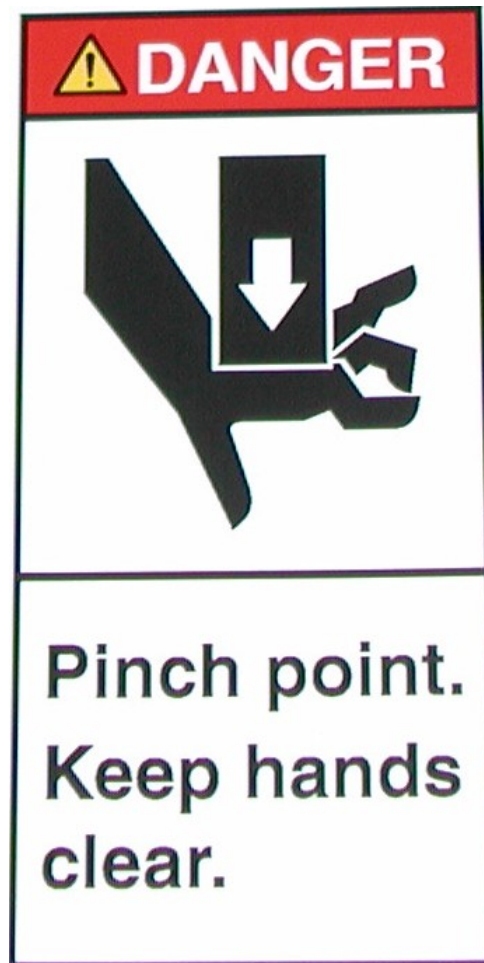
	! WARNING Enclosure is TOP-HEAVY • Stabilize enclosure before opening doors. • Follow guidelines in "Large Enclosure Handling Manual".
--	--

- **DO NOT** remove this label until enclosure is permanently installed.
- Label must be reattached if removed for painting.
- To get additional copies of the "Large Enclosure Handling Manual" or labels, call 1-800-355-3560.

93268001, Dwg. Rev. G

Dangers

Danger labels indicate the probability of death or serious injury and usually show the word DANGER in white writing against a red background. These labels are normally located in areas where serious injury could occur, such as the knife cartridge.



Notices

Notice labels indicate the potential for property damage and usually show the word NOTICE in white writing against a blue backdrop. This label is normally located at the front of the machine.



Serial Number

The serial number label indicates the name and contact information for the company, specific serial number for the machine, and voltage and amp information for the operating system. This specific information relates to the table only. Each auxiliary unit has its own serial number tag with machine-specific information.



[Introduction](#)

Innovation. Quality. Performance. Best in Industry. Worldwide.

Fast Starts

[top](#)

Fast Start ATC Spindle	2
Fast Start MultiVision – ATC Spindle	5
Fast Start Manual Spindle	9
Fast Start MultiVision – Manual Spindle	12
Fast Start Cartridge Knife – ATC Spindle	15
Fast Start Cartridge Knife – Manual Spindle	19
Fast Start Cartridge Knife – MultiVision	23
Fast Start Cartridge Knife System	26
Squaring the Table	29
Hints and Tips	32
Contacting MultiCam	34
Maintenance	35
Maintenance Log	36
Computerized Routing System Warranty	37
Warranty Registration List	38

Fast Start – ATC Spindle

1. Turn on Power

Locate the power switch, the round knob in the control enclosure, and rotate it to the ON position.

Turn on the air regulator.

2. Find Machine Hard Home

Press **Shift**  and **Go Home**  on the keypad to index the gantry, carriage, and Z assembly.

Press **Shift**  and **Five**  to open the chuck and remove a tool from the Tool Changer.

3. Calibrate New Tools

Press **Menu** .

Press **X-axis Jog Arrows**   until ATC is highlighted in blue.

Press **Enter**  to access the ATC sub-menu.

Press **X-axis Jog Arrows**   until Cal_Tool is highlighted in blue.

Press **Enter**  to access the Tool Calibration function.

Select the number of the tool to calibrate using the number keypad (e.g., 1).

Press **Enter** . The spindle will move over to the tool carousel, unload the tool if one is currently in place, rotate to the selected tool, and load that tool into the system. The system will then move over the calibration block.

Press the **Z-axis Jog Arrows**   to lower the spindle until it is near the top of the calibration block. Place the grounding cradle on the ATC mounting groove in the spindle tool holder.

Press and hold **Zero**  to begin the tool calibration. The Z-axis will move down slowly until it senses the calibration block. It will then lift off the block up to maximum Z height. Remove the mounting collar before beginning work.

MultiCam requires that tools be calibrated when a new tool is inserted into the tool holder. The machine will store the Z-axis tool length of the new tool in memory even after the machine is turned off. Only new tools need to be calibrated.

4. Set Home

Press **Jog Arrows**  to place the **center of the bit** over the front right corner of the material.

Press **Jog Speed**  to set the speed of the movement to slow, medium, or fast as necessary for the project and material.

Press **Set Home**  to set the home location for the bit. The system will prompt the operator as to whether the selected position should be set as home, and the display will read “Press Down to set Rotation Point.” See Rotate File for a description of this function.

Press **Enter**  to confirm the position.

5. Turn on the Table Vacuum

Locate the power switch on the table vacuum.

Switch the indicator to the ON position.

The Table Vacuum must be on so that the **surface and maximum depth** can be set properly. Failure to turn on the vacuum will allow an incorrect setting to be entered when using these **MultiCam** features.

6. Set Surface

Press **Jog Arrows**  to position the tool bit anywhere over the material to be cut. Place the grounding collar on the ATC mounting groove in the spindle tool holder.

Press **Set Surface**  and enter the tool number. Place the surface block on top of the material underneath the tool.

Press **Z-axis Jog Arrows**  to lower the Z-axis until the end of the tool is just above the surface block.

Press and hold **Zero**  to begin the surfacing routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height.

Remove the grounding collar and surface block before beginning work.

7. Set Maximum Depth (as needed)

Press **Jog Arrows**  to position the tool over the spoilboard or sacrifice material. Place the grounding cradle on the ATC mounting groove in the spindle tool holder.

Press **Set Max Depth**  to set the maximum depth for the bit.

Place the surface block underneath the tool and on top of the tabletop or at the required maximum depth.

Press **Z-axis Jog Arrows**  to lower the spindle until the end of the tool is just above the surface block.

Press and hold **Zero**  to begin the Maximum Depth routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height.

Remove the grounding collar and surface block before beginning work.

8. Adjust Maximum Depth (as needed)

Press **Shift**  and **Set Max Depth** .

Press **Cancel**  to clear the current maximum depth without having to reboot the machine or press

Z-axis Jog Arrows   to adjust maximum depth in one-thousandth increments (e.g., 0.001). A positive value will increase the range of maximum depth, and a negative value will decrease the range of maximum depth.

Press **Enter**  to accept or **Cancel**  to abort.

9. Turn on Mister or Dust Collection Vacuum

Make sure the manual switch on the mister is switched to the ON position if applicable.

10. DNC

Make sure that JobNameServer  has been activated on the computer.

Press **DNC** . The keypad will display a list of job folders and job files, which are located in the C:\Dncfiles directory.

Press **X-axis Jog Arrows**   until the appropriate folder is highlighted in blue.

Press **Enter**  to display the DNC files that are available in that particular folder.

Press **X-axis Jog Arrows**   until the appropriate file is highlighted in blue.

Press **Start**  to begin the file.

11. Start / Cancel / Pause

Press **Start**  to begin the cutting sequence.

Press **Cancel**  to abort the cutting sequence.

Press **Pause**  to yield the cutting sequence.

Operators should never leave the machine unattended during the cutting sequence.

[Fast Starts](#)

Fast Start MultiVision – ATC Spindle

1. Turn on Power

Locate the power switch, the round knob in the control enclosure, and rotate it to the ON position.

Turn on the air regulator.

Make sure that Suite4 and VisionTool4 or MultiVision are installed on the computer

Make sure the dongle is in place on the computer as the software will not run without the dongle.

2. Find Machine Hard Home

Press **Shift**  and **Go Home**  to index the gantry, carriage, and Z assembly.

Press **Shift**  and **Five**  to open the chuck and remove a tool from the Tool Changer.

3. Calibrate New Tools

Press **Menu** .

Press **X-axis Jog Arrows**   until ATC is highlighted in blue.

Press **Enter**  to access the ATC sub-menu.

Press **X-axis Jog Arrows**   until Calibrate Tool is highlighted in blue.

Press **Enter**  to access the Tool Calibration function.

Select the number of the tool to calibrate using the number keypad (e.g., 1).

Press **Enter** . The spindle will move over to the tool carousel, unload the tool if one is currently in place, rotate to the selected tool, and load that tool into the system. The system will then move over the calibration block.

Press the **Z-axis Jog Arrows**   to lower the spindle until it is near the top of the calibration block. Place the grounding cradle on the ATC mounting groove in the spindle tool holder.

Press and hold **Zero**  to begin the tool calibration. The Z-axis will move down slowly until it senses the calibration block. It will then lift off the block up to maximum Z height. Remove the mounting collar before beginning work.

MultiCam requires that tools be calibrated when a new tool is inserted into the tool holder. The machine will store the Z-axis tool length of the new tool in memory even after the machine is turned off.

Only new tools need to be calibrated.

4. Turn on Table Vacuum

Locate the power switch on the table vacuum.

Switch the indicator to the ON position.

The table vacuum must be on so that the surface can be set properly. Failure to turn on the table vacuum will skew the configuration, which could affect the quality of the cutting sequence.

5. Focus the Camera after Powering Up the Machine

Press **Menu**  to access the menu system.

Press **X-axis Jog Arrows**  until Vision is highlighted in blue.

Press **Enter**  to access the Vision sub-menu.

Press **X-axis Jog Arrows**  until Focus Camera is highlighted in blue.

Press **Enter**  to access the Focus Camera function. The system will prompt the operator to unload the tool.

Press **Enter**  to unload the tool or **Cancel**  to leave the tool in the spindle.

Press **Jog Arrows**  to reach the fiducial on the material to be cut.

Press **Z-axis Jog Arrows**  to focus the camera on the fiducial.

Press **Enter**  when the fiducial is in focus.

6. Set the Surface to the Top of the Material

Use any calibrated tool.

Place the grounding collar on the ATC mounting groove in the spindle tool holder.

Press **Set Surface**  and enter the tool number.

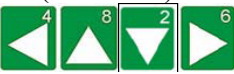
Place the surface block on top of the material underneath the tool.

Press **Z-axis Jog Arrows**  to lower the spindle until the bit is just above the surface block. The operator should monitor the Jog speed when lowering the Z-axis as the machine will only recognize the surface block when 0 is pressed. The spindle could crash into the material or the surface block and break the bit.

Press and hold **Zero**  to begin the surfacing routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height.

Remove the grounding collar/magnet and surface block before beginning work.

7. Set Maximum Depth (as needed)

Press **Jog Arrows**  to position the tool over the spoilboard or sacrifice material.
Place the grounding cradle/magnet on the ATC mounting groove in the spindle tool holder.

Press **Set Max Depth**  to set the maximum depth for the bit.
Place the surface block underneath the tool and on top of the tabletop or at the required maximum depth.

Press **Z-axis Jog Arrows**  to lower the spindle until the end of the tool is just above the surface block.

Press and hold **Zero**  to begin the Maximum Depth routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height.
Remove the grounding collar/magnet and surface block before beginning work.

8. Adjust Maximum Depth (as needed)

Press **Shift**  and **Set Max Depth** .

Press **Cancel**  to clear the current maximum depth without having to reboot the machine or press

Z-axis Jog Arrows  to adjust maximum depth in one-thousandth increments (e.g., 0.001). A positive value will increase the range of maximum depth, and a negative value will decrease the range of maximum depth.

Press **Enter**  to accept or **Cancel**  to abort.

9. Set Home

Press **Jog Arrows**  to place the center of the bit over the first fiducial.

Press **Set Home** .

Press **Enter**  to accept the location.

10. Access a MultiVision Job File

Press **DNC**  and select the file. The VisionTool application will become active and the camera will Jog to the estimated location for the first fiducial.

Watch the system Jog to the second fiducial once the camera locates the first fiducial. The value entered in Auto Find Setting will determine how far the camera will search for the second fiducial.

Watch the system interpolate the exact location of the third fiducial and scan it.

11. Start / Cancel / Pause

Press **Start**  to begin the cutting sequence.

Press **Cancel**  to abort the cutting sequence.

Press **Pause**  to yield the cutting sequence.

Operators should never leave the machine unattended during the cutting sequence.

[Fast Starts](#)

Fast Start – Manual Spindle

1. Turn on Power

Locate the power switch, the round knob in the control enclosure, and rotate it to the ON position.

Turn on the air regulator.

2. Find Machine Hard Home

Press **Shift**  and **Go Home**  on the keypad to index the gantry, carriage, and Z assembly. The activation of this function enables the operator to access the spindle inverter.

3. Insert Cutter

Insert the tool into the collet.

Twist the tool to lock it into place.

4. Set Home

Press **Jog Arrows**  to place the **center of the bit** over the front right corner of the material.

Press **Jog Speed**  to set the speed of the movement to slow, medium, or fast as necessary for the project and material.

Press **Set Home**  to set the home location for the bit. The system will prompt the operator as to whether the selected position should be set as home, and the display will read “Press Down to set Rotation Point.” See Rotate File for a description of this function.

Press **Enter**  to confirm the position.

5. Turn on Table Vacuum

Locate the power switch on the table vacuum.

Switch the indicator to the ON position.

The Table Vacuum must be on so that the **surface and maximum depth** can be set properly. Failure to turn on the vacuum will allow an incorrect setting to be entered when using these **MultiCam** features.

6. Set Surface

Press **Set Surface** .
Place the surface block on top of the material underneath the tool.

Press **Z-axis Jog Arrows**  to lower the Z-axis until the end of the tool is just above the surface block.

Press and hold **Zero**  to begin the surfacing routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height.

Remove the surface block before beginning work.

7. Set Maximum Depth (as needed)

Press **Jog Arrows**  to position the tool over the spoilboard or sacrifice material.

Press **Set Max Depth**  to set the maximum depth for the bit.
Place the surface block underneath the tool and on top of the tabletop or at the required maximum depth.

Press **Z-axis Jog Arrows**  to lower the spindle until the end of the tool is just above the surface block.

Press and hold **Zero**  to begin the Maximum Depth routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height.
Remove the surface block before beginning work.

8. Adjust Maximum Depth (as needed)

Press **Shift**  and **Set Max Depth** .

Press **Cancel**  to clear the current maximum depth without having to reboot the machine or press

Z-axis Jog Arrows  to adjust maximum depth in one-thousandth increments (e.g., 0.001). A positive value will increase the range of maximum depth, and a negative value will decrease the range of maximum depth.

Press **Enter**  to accept or **Cancel**  to abort.

9. Turn on Mister or Dust Collection Vacuum

Verify the manual switch on the mister is switched to the ON position if applicable.

10. DNC

Make sure that JobNameServer  has been activated on the computer.

Press **DNC** . The keypad will display a list of job folders and job files, which are located in the C:\Dncfiles directory.

Press **X-axis Jog Arrows**  until the appropriate folder is highlighted in blue.

Press **Enter**  to display the DNC files that are available in that particular folder.

Press **X-axis Jog Arrows**  until the appropriate file is highlighted in blue.

Press **Start**  to begin the file.

11. Start / Cancel / Pause

Press **Start**  to begin the cutting sequence.

Press **Cancel**  to abort the cutting sequence.

Press **Pause**  to yield the cutting sequence.

Operators should never leave the machine unattended during the cutting sequence.

[Fast Starts](#)

Fast Start MultiVision – Manual Spindle

1. Turn on Power

Locate the power switch, the round knob in the control enclosure, and rotate it to the ON position.

Turn on the air regulator.

Make sure that Suite 4 and VisionTool4 or MultiVision are installed on the computer

Make sure the dongle is in place on the computer as the software will not run without the dongle.

2. Find Machine Hard Home

Press **Shift**  and **Go Home**  on the keypad to index the gantry, carriage, and Z assembly. The activation of this function enables the operator to access the spindle inverter.

3. Insert Cutter

Insert the tool into the collet.

Twist the tool to lock it into place.

4. Turn on Table Vacuum

Locate the power switch on the table vacuum.

Switch the indicator to the ON position.

The table vacuum must be on so that the surface and maximum depth can be set properly. Failure to turn on the table vacuum will skew the configuration, which could affect the quality of the cutting sequence.

5. Set the Surface to the Top of the Material

Press **Set Surface** .

Place the surface block on top of the material underneath the tool.

Press **Z-axis Jog Arrows**   to lower the spindle until the bit is just above the surface block. The operator should monitor the Jog speed when lowering the Z-axis as the machine will only recognize the surface block when 0 is pressed. The spindle could crash into the material or the surface block and break the bit.

Press and hold **Zero**  to begin the surfacing routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height.

Remove the surface block before beginning work.

6. Set Maximum Depth (as needed)

Press **Jog Arrows**  to position the tool over the spoilboard or sacrifice material.

Press **Set Max Depth**  to set the maximum depth for the bit.

Place the surface block underneath the tool and on top of the tabletop or at the required maximum depth.

Press **Z-axis Jog Arrows**  to lower the spindle until the end of the tool is just above the surface block.

Press and hold **Zero**  to begin the Maximum Depth routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height.

Remove the surface block before beginning work.

7. Adjust Maximum Depth (as needed)

Press **Shift**  and **Set Max Depth** .

Press **Cancel**  to clear the current maximum depth without having to reboot the machine or press

Z-axis Jog Arrows  to adjust maximum depth in one-thousandth increments (e.g., 0.001). A positive value will increase the range of maximum depth, and a negative value will decrease the range of maximum depth.

Press **Enter**  to accept or **Cancel**  to abort.

8. Set Home

Press **Jog Arrows**  to place the center of the bit over the first fiducial.

Press **Set Home** .

Press **Enter**  to accept the location.

9. Focus the Camera Each Time the Material Thickness Changes

Press **Menu**  to access the menu system.

Press **X-axis Jog Arrows**  until Vision is highlighted in blue.

Press **Enter**  to access the Vision sub-menu.

Press **X-axis Jog Arrows**  until Focus Camera is highlighted in blue.

Press **Enter**  to access the Focus Camera function. The system will prompt the operator to unload the tool.

Press **Enter**  to unload the tool or **Cancel**  to leave the tool in the spindle.

Press **Jog Arrows**  to reach the fiducial on the material to be cut.

Press **Z-axis Jog Arrows**  to focus the camera on the fiducial.

Press **Enter**  when the fiducial is in focus.

10. Access a MultiVision Job File

Press **DNC**  and select the file. The VisionTool application will become active and the camera will Jog to the estimated location for the first fiducial.

Watch the system Jog to the second fiducial once the camera locates the first fiducial. The value entered in Auto Find Setting will determine how far the camera will search for the second fiducial.

Watch the system interpolate the exact location of the third fiducial and scan it.

11. Start / Cancel / Pause

Press **Start**  to begin the cutting sequence.

Press **Cancel**  to abort the cutting sequence.

Press **Pause**  to yield the cutting sequence.

Operators should never leave the machine unattended during the cutting sequence.

[Fast Starts](#)

Fast Start Cartridge Knife – ATC Spindle

1. Turn on Power

Locate the power switch, the round knob in the control enclosure, and rotate it to the ON position.

Turn on the air regulator.

2. Find Machine Hard Home with the Router

Press **Shift**  and **Go Home**  on the keypad to index the gantry, carriage, and Z assembly.

Press **Shift**  and **Five**  to open the chuck and remove a tool from the Tool Changer.

3. Calibrate New Tools (including Cartridges 31 – 40 and 41 – 50 if applicable)

Press **Menu** .

Press **X-axis Jog Arrows**  until ATC is highlighted in blue.

Press **Enter**  to access the ATC sub-menu.

Press **X-axis Jog Arrows**  until Cal_Tool is highlighted in blue.

Press **Enter**  to access the Tool Calibration function.

Select the number of the tool to calibrate using the number keypad (e.g., 1).

Press **Enter** . The spindle will move over to the tool carousel, unload the tool if one is currently in place, rotate to the selected tool, and load that tool into the system. The system will then move over the calibration block.

Press the **Z-axis Jog Arrows**  to lower the spindle until it is near the top of the calibration block. Place the grounding cradle on the ATC mounting groove in the spindle tool holder.

Press and hold **Zero**  to begin the tool calibration. The Z-axis will move down slowly until it senses the calibration block. It will then lift off the block up to maximum Z height.

Remove the mounting collar before beginning work.

MultiCam requires that tools be calibrated when a new tool is inserted into the tool holder. The machine will store the Z-axis tool length of the new tool in memory even after the machine is turned off.

Only new tools need to be calibrated.

4. Turn on the Table Vacuum

Locate the power switch on the table vacuum.

Switch the indicator to the ON position.

The Table Vacuum must be on so that the **surface and maximum depth** can be set properly. Failure to turn on the vacuum will allow an incorrect setting to be entered when using these **MultiCam** features.

5. Set Surface with the Router (must be reset each time the machine is turned on, the bit changes, or the material thickness changes)

Press **Jog Arrows**  to position any calibrated tool anywhere over the surface block. If operators are calibrating the knife, then they should set surface with Tool 1.
Place the grounding collar on the ATC mounting groove in the spindle tool holder.

Press **Set Surface**  and enter the tool number.
Place the surface block on top of the material underneath the tool.

Press **Z-axis Jog Arrows**  to lower the Z-axis until the end of the tool is just above the surface block.

Press and hold **Zero**  to begin the surfacing routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height.

6. Set Maximum Depth for the Router (as needed and then after each reboot)

Press **Jog Arrows**  to position the tool over the spoilboard or sacrifice material.
Place the grounding cradle on the ATC mounting groove in the spindle tool holder.

Press **Set Max Depth**  to set the maximum depth for the bit.

Place the surface block underneath the tool and on top of the tabletop or at the required maximum depth.

Press **Z-axis Jog Arrows**  to lower the spindle until the end of the tool is just above the surface block.

Press and hold **Zero**  to begin the Maximum Depth routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height. **Maximum Depth** has no effect on the knife.
Remove the grounding collar and surface block before beginning work.

7. Adjust Maximum Depth for the Router (as needed)

Press **Shift**  and **Set Max Depth** .

Press **Cancel**  to clear the current maximum depth without having to reboot the machine or press

Z-axis Jog Arrows  to adjust maximum depth in one-thousandth increments (e.g., 0.001). A positive value will increase the range of maximum depth, and a negative value will decrease the range of maximum depth.

Press **Enter**  to accept or **Cancel**  to abort. **Adjust Maximum Depth** has no effect on the knife.

8. Set Home with the Router

Press **Jog Arrows**  to place the **center of the bit** over the front right corner of the material.

Press **Jog Speed**  to set the speed of the movement to slow, medium, or fast as necessary for the project and material.

Press **Set Home**  to set the home location for the bit. The system will prompt the operator as to whether the selected position should be set as home, and the display will read “Press Down to set Rotation Point.” See Rotate File for a description of this function.

Press **Enter**  to confirm the position.

9. Set Knife Params

Keypad

Press **Menu** .

Press **X-axis Jog Arrows**  until KnifeUtil is highlighted in blue.

Press **Enter**  to access the Knife Utility sub-menu.

Press **X-axis Jog Arrows**  until Knife Control is highlighted in blue.

Press **Enter**  to toggle to Keypad.

Press **Cancel**  to exit the KnifeUtil sub-menu.

Press **X-axis Jog Arrows**  until Knife is highlighted in blue.

Press **Enter**  to access the Knife sub-menu.

Press **X-axis Jog Arrows**  until Knife Depth is highlighted in blue.

Press **Enter**  to set the knife depth.

Enter a positive or negative value for the knife depth and press **Enter**  to return to the Knife sub-menu.

Press **X-axis Jog Arrows**  until Knife Lift is highlighted in blue.

Press **Enter**  to set the knife lift.

Enter the value for the knife lift and press **Enter**  to return to the Knife sub-menu.

Job File

Press **Menu** .

Press **X-axis Jog Arrows**   until KnifeUtil is highlighted in blue.

Press **Enter**  to access the Knife Utility sub-menu.

Press **X-axis Jog Arrows**   until Knife Control is highlighted in blue.

Press **Enter**  to toggle to Job.

Skip to 10. **Turn on Mister or Dust Collection Vacuum** below.

10. Turn on Mister or Dust Collection Vacuum

Make sure the manual switch on the mister is switched to the ON position if applicable.

11. DNC

Make sure that JobNameServer  has been activated on the computer.

Press **DNC** . The keypad will display a list of job folders and job files, which are located in the C:\Dncfiles directory.

Press **X-axis Jog Arrows**   until the appropriate folder is highlighted in blue.

Press **Enter**  to display the DNC files that are available in that particular folder.

Press **X-axis Jog Arrows**   until the appropriate file is highlighted in blue.

Press **Start**  to begin the file.

12. Start / Cancel / Pause

Press **Start**  to begin the cutting sequence.

Press **Cancel**  to abort the cutting sequence.

Press **Pause**  to yield the cutting sequence.

Operators should never leave the machine unattended during the cutting sequence.

[Fast Starts](#)

Fast Start Cartridge Knife – Manual Spindle

1. Turn on Power

Locate the power switch, the round knob in the control enclosure, and rotate it to the ON position.

Turn on the air regulator.

2. Find Machine Hard Home with the Router

Press **Shift**  and **Go Home**  on the keypad to index the gantry, carriage, and Z assembly.

Press **Shift**  and **Five**  to open the chuck and remove a tool from the Tool Changer.

3. Insert Cutter

Insert the tool into the collet.

Twist the tool to lock it into place.

4. Turn on the Table Vacuum

Locate the power switch on the table vacuum.

Switch the indicator to the ON position.

The Table Vacuum must be on so that the **surface and maximum depth** can be set properly. Failure to turn on the vacuum will allow an incorrect setting to be entered when using these **MultiCam** features.

5. Set Home with the Router

Press **Jog Arrows**  to place the **center of the bit** over the front right corner of the material.

Press **Jog Speed**  to set the speed of the movement to slow, medium, or fast as necessary for the project and material.

Press **Set Home**  to set the home location for the bit. The system will prompt the operator as to whether the selected position should be set as home, and the display will read “Press Down to set Rotation Point.” See Rotate File for a description of this function.

Press **Enter**  to confirm the position.

6. Set Surface with the Router

Press **Set Surface**  and enter any regular tool number (e.g., 1).
Place the surface block on top of the material underneath the tool.

Press **Z-axis Jog Arrows**  to lower the Z-axis until the end of the tool is just above the surface block.

Press and hold **Zero**  to begin the surfacing routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height.

Remove the surface block before beginning work.

7. Set Surface with the Knife

Press **Set Surface**  and enter any knife tool number (e.g., 31).

Press **Cancel**  to leave oscillations inactive. Oscillation stroke can be added in later.
Place the surface block on top of the material underneath the tool.

Press **Z-axis Jog Arrows**   to lower the Z-axis until the end of the knife is just above the surface block.

Press and hold **Zero**  to begin the surfacing routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height.

Press **Enter**  to add in oscillating stroke or **Cancel**  to exclude oscillating stroke.
Remove the surface block before beginning work.

8. Set Maximum Depth for the Router (as needed and then after each reboot)

Press **Jog Arrows**     to position the tool over the spoilboard or sacrifice material.
Place the grounding cradle on the ATC mounting groove in the spindle tool holder.

Press **Set Max Depth**  to set the maximum depth for the bit.

Place the surface block underneath the tool and on top of the tabletop or at the required maximum depth.

Press **Z-axis Jog Arrows**   to lower the spindle until the end of the tool is just above the surface block.

Press and hold **Zero**  to begin the Maximum Depth routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height. **Maximum Depth has no effect on the knife.**
Remove the grounding collar and surface block before beginning work.

9. Adjust Maximum Depth for the Router (as needed)

Press **Shift**  and **Set Max Depth** .

Press **Cancel**  to clear the current maximum depth without having to reboot the machine or press

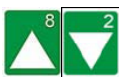
Z-axis Jog Arrows   to adjust maximum depth in one-thousandth increments (e.g., 0.001). A positive value will increase the range of maximum depth, and a negative value will decrease the range of maximum depth.

Press **Enter**  to accept or **Cancel**  to abort. **Adjust Maximum Depth has no effect on the knife.**

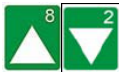
10. Set Knife Params

Job File

Press **Menu** .

Press **X-axis Jog Arrows**  until KnifeUtil is highlighted in blue.

Press **Enter**  to access the Knife Utility sub-menu.

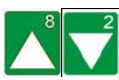
Press **X-axis Jog Arrows**  until Knife Control is highlighted in blue.

Press **Enter**  to toggle to Job.

Skip to 10. **Turn on Mister or Dust Collection Vacuum** below.

Keypad

Press **Menu** .

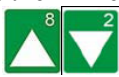
Press **X-axis Jog Arrows**  until KnifeUtil is highlighted in blue.

Press **Enter**  to access the Knife Utility sub-menu.

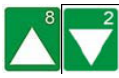
Press **X-axis Jog Arrows**  until Knife Control is highlighted in blue.

Press **Enter**  to toggle to Keypad.

Press **Cancel**  to exit the KnifeUtil sub-menu.

Press **X-axis Jog Arrows**  until Knife is highlighted in blue.

Press **Enter**  to access the Knife sub-menu.

Press **X-axis Jog Arrows**  until Knife Depth is highlighted in blue.

Press **Enter**  to set the knife depth.

Enter a positive or negative value for the knife depth and press **Enter**  to return to the Knife sub-menu.

Press **X-axis Jog Arrows**  until Knife Lift is highlighted in blue.

Press **Enter**  to set the knife lift.

Enter the value for the knife lift and press **Enter**  to return to the Knife sub-menu.

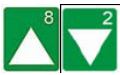
11. Turn on Mister or Dust Collection Vacuum

Make sure the manual switch on the mister is switched to the ON position if applicable.

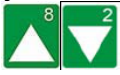
12. DNC

Make sure that JobNameServer  has been activated on the computer.

Press **DNC** . The keypad will display a list of job folders and job files, which are located in the C:\Dncfiles directory.

Press **X-axis Jog Arrows**  until the appropriate folder is highlighted in blue.

Press **Enter**  to display the DNC files that are available in that particular folder.

Press **X-axis Jog Arrows**  until the appropriate file is highlighted in blue.

Press **Start**  to begin the file.

13. Start / Cancel / Pause

Press **Start**  to begin the cutting sequence.

Press **Cancel**  to abort the cutting sequence.

Press **Pause**  to yield the cutting sequence.

Operators should never leave the machine unattended during the cutting sequence.

[Fast Starts](#)

Fast Start Cartridge Knife – MultiVision

1. Turn on Power

Locate the power switch, the round knob in the control enclosure, and rotate it to the ON position.
Make sure that Suite 4 and VisionTool4 or MultiVision are installed on the computer
Make sure the dongle is in place on the computer as the software will not run without the dongle.

2. Find Machine Hard Home

Press **Shift**  and **Go Home**  to index the gantry, carriage, and Z assembly.

3. Insert Cartridge

Insert the cartridge into the cartridge receiver.
Tighten the 3 bolts. The cartridge will only fit inside the cartridge receiver one way.

4. Turn on Table Vacuum

Locate the power switch on the table vacuum.
Switch the indicator to the ON position.
The Table Vacuum must be on so that the surface and depth can be set properly. Failure to turn on the vacuum will skew the configuration, which could affect the quality of the cutting sequence.

5. Set Home

Press **Jog Arrows**  to place the **center of the blade** over the front right corner of the material.

Press **Jog Speed**  to set the speed of the movement to slow, medium, or fast as necessary for the project and material.

Press **Set Home**  to set the home location. The system will prompt the operator as to whether the selected position should be set as home, and the display will read “Press Down to set Rotation Point.” See Rotate File for a description of this function.

Press **Enter**  to confirm the position.

6. Set Knife Params

Press **Menu** .

Press **X-axis Jog Arrows**  until KnifeUtil is highlighted in blue.

Press **Enter**  to access the Knife Utility sub-menu.

Press **X-axis Jog Arrows**  until Knife Control is highlighted in blue.

Press **Enter**  to toggle to Keypad.

Press **Cancel**  to exit the KnifeUtil sub-menu.

Press **X-axis Jog Arrows**  until Knife is highlighted in blue.

Press **Enter**  to access the Knife sub-menu.

Press **X-axis Jog Arrows**  until Knife Depth is highlighted in blue.

Press **Enter**  to set the knife depth.

Enter a positive or negative value for the knife depth and press **Enter**  to return to the Knife sub-menu.

Press **X-axis Jog Arrows**  until Knife Lift is highlighted in blue.

Press **Enter**  to set the knife lift.

Enter the value for the knife lift and press **Enter**  to return to the Knife sub-menu.

7. Set Surface with the Knife (must be reset each time the machine is turned on, the blade changes, or the material thickness changes)

Press **Set Surface**  and enter any knife tool number (e.g., 31).

Press **Cancel**  to leave oscillations inactive. Oscillation stroke can be added in later.
Place the surface block on top of the material underneath the tool.

Press **Z-axis Jog Arrows**  to lower the Z-axis until the end of the knife is just above the surface block.
On Solenoid-controlled systems, the surface must be set to the bottom of the material.

Press and hold **Zero**  to begin the surfacing routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Knife Lift Height.

Press **Enter**  to add in oscillating stroke or **Cancel**  to exclude oscillating stroke.
Remove the surface block before beginning work.

8. Capture the Fiducial

Click on Live Feed  in the toolbar and Jog the camera over a fiducial. Operators may need to move the camera along the Z-axis to bring the fiducial into focus, and the spotlight will flash while the camera is in Live Feed mode.

Click on Capture Fiducial  in the toolbar. A box should appear around the fiducial .

9. Focus the Camera Each Time the Material Thickness Changes

Press **Menu** .

Press **X-axis Jog Arrows**  until Vision is highlighted in blue.

Press **Enter**  to access the Vision sub-menu.

Press **X-axis Jog Arrows**  until Focus Camera is highlighted in blue.

Press **Enter**  to access the Focus Camera function.

Press **Jog Arrows**  to reach the fiducial on the material to be cut.

Press **Z-axis Jog Arrows**  to focus the camera on the fiducial.

Press **Enter**  when the fiducial is in focus.

10. Access a MultiVision Job File

Make sure that JobNameServer  has been activated on the computer.

Press **DNC**  and select the file. The VisionTool application will become active and the camera will Jog to the estimated location for the first fiducial.

Watch the system Jog to the second fiducial once the camera locates the first fiducial. The value entered in Auto Find Setting will determine how far the camera will search for the second fiducial.

Watch the system interpolate the exact location of the third fiducial and scan it.

11. Start / Cancel / Pause

Press **Start**  to begin the cutting sequence.

Press **Cancel**  to abort the cutting sequence.

Press **Pause**  to yield the cutting sequence.

Operators should never leave the machine unattended during the cutting sequence.

[Fast Starts](#)

Fast Start Cartridge Knife System

1. Turn on Power

Locate the power switch, the round knob in the control enclosure, and rotate it to the ON position.

Turn on the air regulator.

2. Find Machine Hard Home

Press **Shift**  and **Go Home**  on the keypad to index the gantry, carriage, and Z assembly.

Press **Shift**  and **Five**  to open the chuck and remove a tool from the Tool Changer.

3. Insert Cartridge

Insert the cartridge into the cartridge receiver.

Tighten the 3 bolts. The cartridge will only fit inside the cartridge receiver one way.

4. Turn on the Table Vacuum

Locate the power switch on the table vacuum.

Switch the indicator to the ON position.

The Table Vacuum must be on so that the **surface and depth** can be set properly. Failure to turn on the vacuum will allow an incorrect setting to be entered when using these **MultiCam** features.

5. Set Home

Press **Jog Arrows**  to place the **center of the blade** over the front right corner of the material.

Press **Jog Speed**  to set the speed of the movement to slow, medium, or fast as necessary for the project and material.

Press **Set Home**  to set the home location. The system will prompt the operator as to whether the selected position should be set as home, and the display will read "Press Down to set Rotation Point." See Rotate File for a description of this function.

Press **Enter**  to confirm the position.

6. Set Knife Params

Press **Menu** .

Press **X-axis Jog Arrows**  until KnifeUtil is highlighted in blue.

Press **Enter**  to access the Knife Utility sub-menu.

Press **X-axis Jog Arrows**  until Knife Control is highlighted in blue.

Press **Enter**  to toggle to Keypad.

Press **Cancel**  to exit the KnifeUtil sub-menu.

Press **X-axis Jog Arrows**  until Knife is highlighted in blue.

Press **Enter**  to access the Knife sub-menu.

Press **X-axis Jog Arrows**  until Knife Depth is highlighted in blue.

Press **Enter**  to set the knife depth.

Enter a positive or negative value for the knife depth and press **Enter**  to return to the Knife sub-menu.

Press **X-axis Jog Arrows**  until Knife Lift is highlighted in blue.

Press **Enter**  to set the knife lift.

Enter the value for the knife lift and press **Enter**  to return to the Knife sub-menu.

7. Set Surface with the Knife (must be reset each time the machine is turned on, the blade changes, or the material thickness changes)

Press **Set Surface**  and enter any knife tool number (e.g., 31).

Press **Cancel**  to leave oscillations inactive. Oscillation stroke can be added in later.
Place the surface block on top of the material underneath the tool.

Press **Z-axis Jog Arrows**  to lower the Z-axis until the end of the knife is just above the surface block.

Press and hold **Zero**  to begin the surfacing routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Knife Lift Height.

Press **Enter**  to add in oscillating stroke or **Cancel**  to exclude oscillating stroke.
Remove the surface block before beginning work.

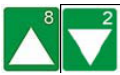
8. Turn on Mister or Dust Collection Vacuum

Make sure the manual switch on the mister is switched to the ON position if applicable.

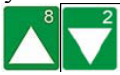
9. DNC

Make sure that JobNameServer  has been activated on the computer.

Press **DNC** . The keypad will display a list of job folders and job files, which are located in the C:\Dncfiles directory.

Press **X-axis Jog Arrows**  until the appropriate folder is highlighted in blue.

Press **Enter**  to display the DNC files that are available in that particular folder.

Press **X-axis Jog Arrows**  until the appropriate file is highlighted in blue.

Press **Start**  to begin the file.

10. Start / Cancel / Pause

Press **Start**  to begin the cutting sequence.

Press **Cancel**  to abort the cutting sequence.

Press **Pause**  to yield the cutting sequence.

Operators should never leave the machine unattended during the cutting sequence.

[Fast Starts](#)

Squaring the Table

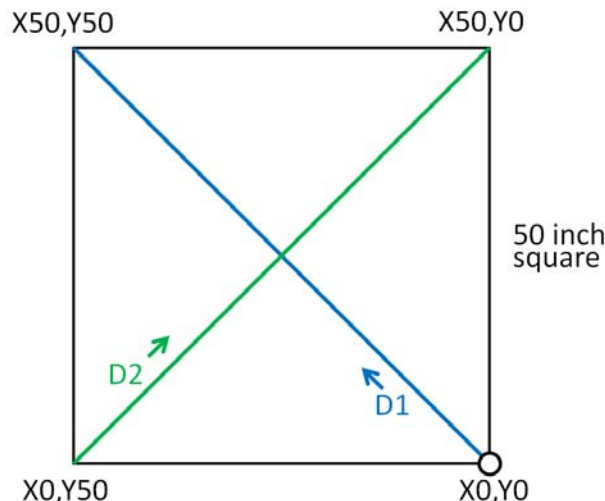
The table squaring procedure should be completed when the **MultiCam** machine is first installed. Operators may need to resquare the table if a forklift hits the gantry or if the motors are changed. The following directions describe how to check the square of the table, change the square of the table, and verify the square of the table. Some helpful hints are also provided.

The required tools are as follows:

- Tape measure
- Cutting head prepared to mark material (e.g., V point cutter, permanent marker)
- Two people
- Calculator

Check the Square of the Table

1. Home the machine.
2. Install a V point cutter or permanent marker to the cutting head.
3. Jog the cutting head to the lower right corner of the table.
4. Press **Set Home** .
5. Jog the cutting head down in Z to make a small mark at X0,Y0.
6. Use the Move command on the keypad to exactly position the cutting head at the next corner of a large square and make another small mark.
7. Repeat step #6 until all 4 corners of the square have been marked [e.g., place marks at (0,0), (50,0), (50,50), (0,50)].
8. Measure the distance between the two diagonal points (D1 and D2) using the tape measure.
 - If the distances are the same, then the machine is square.
 - If the distances are different, then continue with **Change the Square of the Table**.



Change the Square of the Table

If D1 and D2 are different lengths, then the operator should use the following formula to correct the square value.

$$\frac{\text{Rail Distance}}{\text{Square Size}} \times \frac{D1 - D2}{\sqrt{2}} = \Delta X \text{ Square}$$

Rail Distance – center to center distance between gantry casting datums (see **Helpful Hints**)

Square Size – length of the sides of the square

D1 – the first diagonal distance (e.g. 0,0 to 50,50)

D2 – the second diagonal distance (e.g. 0,50 to 50,0)

ΔX Square – the amount added to or subtracted from the current X square value

Example:

$$\frac{62.25}{50.0} \times \frac{70.78 - 70.62}{1.41421} = 0.090$$

The calculated ΔX Square value of 0.090 would be added to the current X square value found in the flash parameter location 23.

Example:

Current X square (Param 23)	-0.050
Calculated ΔX Square	+ 0.090
New value for Param 23	+0.040

Verify the Square of the Table

After changing parameter 23 with the new value, the operator must make some additional changes to insure that the square of the table is correct.

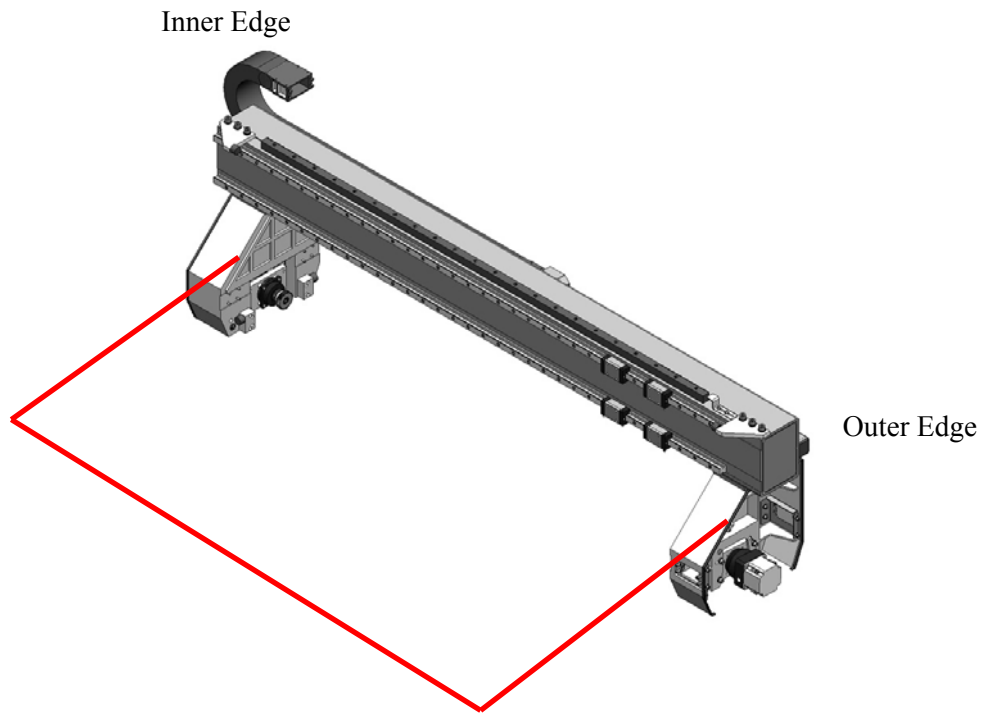
1. Loosen the gantry casting bolts to allow the gantry to twist into its new position on the gantry castings.
2. Reboot the machine.
3. Hard home the machine and repeat **Check the Square of the Table**.

- If the distances are the same, then the machine is square. Operators should tighten the gantry bolts to lock the gantry into place on the gantry castings.

- If the distances are different, then repeat **Change the Square of the Table**.

Helpful Hints

- ❖ At least 2 operators should be involved when checking the square of the table.
- ❖ Operators should begin measuring from the 1" mark on the tape measure. This provides greater accuracy.
- ❖ The measurement will be more accurate if the square is larger (e.g., 50" square is better than 10" square).
- ❖ Accurately squaring the gantry to the machine may take more than one attempt.
- ❖ The rail distance can be measured from the center of the XA gantry casting to the center of the XB gantry casting. Since it is often difficult to locate the exact center of the casting, operators usually start at the outer edge of one gantry casting and measure to the inner edge of the other gantry casting.



[Fast Starts](#)

Hints and Tips

The **MultiCam** systems have a very powerful yet easy-to-use interface. Operators who understand their systems and learn the keypad functions can utilize their cutting systems in an efficient and productive manner. The following tips complement this knowledge and can be used to the operators' advantage. Any operator with additional helpful hints or tips should contact the regional Technology Center so that the User Manual can be updated.



#1 Maintaining Safety

Safety glasses and hearing protection **should be worn at all times** while operating the machine, and long hair should be **restrained with a cap or net** when near the machines. Operators should never leave the machine unattended during the cutting sequence, and labels should be observed at all times.

#2 Testing

When testing new processes or when learning new functions of the machine, use the following 3 procedures:

1. Insert a pen into the end of the spindle and run the file on a piece of paper. Disable the spindle or the pen will spin.
2. Set an imaginary surface above the material high enough that the spindle will still move up and down but never come in contact with the material.
3. Use the Dry Run function.

Do **not** let the spindle run.

#3 Recovering after Loss of Power

When Set Home (0,0)  is pressed, the X,Y locations are stored in the controller's memory. If the system loses power, the job can be restored by returning to the last home and running Proximity Restart.

Power can be applied back to the machine by pressing Find Home   to index the machine. When the machine has completed the find home function, the operator should press Return to Home 0   to move the machine back to the last home position and set home there before re-establishing the Set Surface (Z=0), Cut Depth, Tool Lift, Cut Speed, and Z Plunge Feed. The Proximity Restart function can then be used to resume the file.

#4 Cutting Metal

Single flute spiral up-cutters should be used when cutting aluminum or brass since they allow for faster feedrates and longer cutter life. These cutters will cut out bigger chips, a process which removes heat from the cut and reduces welding. Two (2) flute cutters generally produce a slightly better edge finish than single flute cutters.

Soft metals, such as 30-31 aluminum, should be cut at a high feedrate with single flute straight or spiral cutters.

Three (3) flute Titanium Nitrate (TIN) coated bits should be used for cutting stainless steel.

The Micro Drop Misting System or Coolubricator System should always be used when cutting metals.

#5 Cutting Plastics

The Pressure Foot and the Micro Drop Misting System or Coolubricator System can be very useful tools when cutting plastics. The pressure foot can be used to remove chips and keep the chips from welding back into the cut, and the mister can keep chips from welding back into the cut and help polish the cut.

#6 Maintaining the Machine

The **MultiCam** Router will produce consistent, accurate work as long as it is well-maintained. [Maintenance](#) for this system is easy, yet it should not be overlooked. Operators may refer to the maintenance chart for more detail.

All machines should be kept clean when not in use. The build-up of chips and dust on the system should be cleared daily with special attention focused on the gear rack and bearing rails.

All of the bearings should be greased at least twice a month based on an 8-hour work day. The bearings should be greased more often if the machine is in use more than 8 hours a day.

The Lead Screw Z-axis should be lubricated with a non-aerosol, silicon-based lubricant. Operators are cautioned **against** using the lithium grease provided by **MultiCam** when lubricating Lead Screws as this may lead to premature wear of the Lead Screw. Ball Screws, however, can be lubricated with lithium grease.

The filters on the electronics enclosure and vacuum pump should also be cleaned.

#7 Cutting to Paper Mask

When cutting plastics or paper-masked materials, operators should set the maximum depth to allow for cutting through the material without cutting the paper on the back.

When setting up the system for this application, operators should place a piece of paper under the calibration block. This will set the max depth on top of the paper. [Fast Starts](#)



Innovation. Quality. Performance. Best in Industry. Worldwide.

Contacting **MultiCam**

For Sales questions or concerns, please direct all necessary email correspondence to the following address:

sales@multicam.com

Customers with technical questions about their **MultiCam** equipment should refer to their User Manual first. Any caller with questions regarding an issue that is clearly defined in the manual will be directed back to the manual by the Service Technician.

If the customer is unable to find the answers in the User Manual, he or she may contact the regional Technology Center. When placing a service call, customers should include the model and serial number of the machine and the full name of the company.

[Fast Starts](#)

Maintenance

The **MultiCam** Router will provide years of productive service if it is maintained properly. There are daily, weekly, monthly, quarterly, and yearly maintenance steps required for each machine based on a 40-hour work week. **MultiCam** recommends that any routing system used in excess of 40 hours a week should have the maintenance schedule adjusted accordingly.

Daily

The following tasks should be completed at the beginning of shift daily on the **MultiCam** Router:

- Wipe down rails and clean out racks
- Inspect water lines
- Blow or vacuum any chips off the machine

Weekly

The following tasks should be completed at the end of shift weekly on the **MultiCam** Router:

- Complete all daily tasks
- Clean the machine thoroughly and remove any excess grease grime
- Vacuum the filter on the control box
- Blow out any build-up in the rails or racks
- Remove, clean, and replace the filter or filters on the vacuum hold-down systems
- Check oil levels in the vacuum hold-down system and fill if necessary

Monthly

The following tasks should be completed at the end of last shift monthly on the **MultiCam** Router:

- Complete all weekly tasks
- Grease all the linear bearing cars
- Clean any racks and screws with a scrub brush and degreaser

Quarterly

The following tasks should be completed at the end of last shift quarterly on the **MultiCam** Router:

- Complete all monthly tasks
- Remove X- and Y-axis covers to inspect transmission or gearboxes for wear
- Check the tabletop bolts and tighten if needed

Bi-Annually

The following tasks should be completed at the end of shift bi-annually on the **MultiCam** Router:

- Complete all quarterly tasks
- Turn off power and vacuum out any debris in the electrical cabinet
- Replace belts on all transmissions

[Fast Starts](#)

Computerized Routing System Warranty

MultiCam hereby warrants the electronics, motion controller, and motors of the Computerized Routing System, when properly used per the original design function, to be free of defects for a period of one (1) year from the date of shipment from **MultiCam**. This warranty is provided to assure the initial purchaser that the machine and control units are free from defects in workmanship and materials. No responsibility for loss of machine time or work piece is expressed or implied. This warranty is valid to the original owner while in compliance with normal usage and the recommended maintenance as prescribed by the manufacturer. Failure to perform regularly scheduled maintenance and maintain the maintenance log will void this warranty.

If the electronics, motion controller, or motors of the Routing System are found to be defective during the applicable warranty period in accordance with the following specific procedure, **MultiCam** will, at its option, either repair or replace the parts at no charge. Replacement parts may be refurbished. All parts that are replaced shall become the property of **MultiCam**. This warranty does not include the following items:

vacuum filters	timing belts	hoses
dust brushes	tooling	items consumed in normal use

Also excluded from the warranty is any damage to the Routing System resulting from, but not limited to, shipping, accident, disaster, misuse, abuse, or any unauthorized modification of the product

MultiCam makes no other warranty or representation of any kind, express or implied, and all implied warranties of merchantability and fitness for a particular purpose with respect to the equipment are disclaimed.

Specific Warranties

Spindle Motors

MultiCam will warranty its spindle motors for a period of 3 months from the time of installation. At any time, misuse, abuse, or use of the spindle motor outside of the manufacturer's operating specifications will not be accepted as defective and will void the warranty. After the initial 3 months, **MultiCam** will warrant the spindle motor to be free from defects for a period of 9 months. Spindle bearings are not covered during this 9-month period.

Vacuum Pumps

Positive Displacement systems provided by **MultiCam** have a standard 12-month warranty, which allows for the repair or replacement of parts at no charge. Replacement parts may be refurbished. All parts that are replaced shall become the property of **MultiCam**. Exclusions match those listed above.

Reitschle high-pressure vacuum pumps carry a 12-month warranty covering manufacturing defects.

Limited Warranty Procedures

Inside the USA

Customers must first contact their local authorized **MultiCam** Technology Center to issue a purchase order, which is required for any parts that must be returned to **MultiCam**. When replacement parts are shipped, an RMA number/tag will be included and must be prominently displayed on the outside of the return package as well as on the accompanying packing slip. All domestic parts must be returned within 30 days of the RMA being assigned. If any RMA is received after that time, then the received merchandise will not be accepted for return and will be scrapped or returned at the Distributor's expense. The customer assumes cost of shipment for returned items. **MultiCam** will not issue Call Tags for any machine part or equipment.

Outside the USA

Please contact the local **MultiCam** Distributor for specific instructions.

[Fast Starts](#)

Warranty Registration List

The following documents must be returned to **MultiCam** in order for the warranty to be registered. Some documents may not apply to every **MultiCam** customer.

- * Colombo Spindle Warranty Card
- * EuroSpindle Warranty Card
- * High Pressure Vacuum Warranty Card
- * Positive Displacement Warranty Card
- * Multicam Routing System Warranty Card

These documents should be faxed or mailed to **MultiCam** within 30 days of the machine delivery and installation, with the machine model and serial number, as well as with the company information, included in the correspondence.

Fax Number: (972) 929-4071

Mailing Address:

MultiCam

Attn: Warranty Registration
PO Box 612048
Dallas, TX 75261

[Fast Starts](#)

Keypad and Functions

[top](#)

Handheld Keypad	3
Spindle Warmup	4
Emergency Stop (E-Stop)	5
Hot Keys for the Router	5
Go Home	6
Feedrate Override	7
Jog Z-axis	7
Spindle RPM Override	7
Pause	8
Set Surface	9
Start	10
Set Home	10
Move	11
Jog X- and Y-axes	12
Spindle Test	13
Menu	13
Set Max Depth	14
Shift	14
DNC	15
Bar Code	16
Drive Enable	17
Cancel	17
Jog Speed	17
Help	18
Soft Homes	18
Enter	19
Find Home	19
Park Z Up or Down	19
Spindle RPM	20
Fasmer Surface	21
Spindle Mode	22
Proximity Restart	23
Return to Home 0	26
Park X High or Low	27
Mister Test	27
Adjust Max Depth	28
Park Y High or Low	28
Open Chuck	29
Drive Disable	31
Controller Information	32
Execute Self Test	33
Set Soft Homes	33
Advanced	34



KF

2

Innovation. Quality. Performance. Best in Industry. Worldwide.

<u>Advanced Features</u>	36
<u>Dry Run</u>	36
<u>Rotate File</u>	37
<u>Step and Repeat</u>	38
<u>Dual Start</u>	40

Handheld Keypad

The **MultiCam** graphical keypad provides more flexibility in accessing the Router by allowing the operator to be closer to the work surface when setting up jobs. With the mobility of this unit, operators experience fewer problems pressing wrong keys by maintaining visual contact with the keypad. The display also allows for the standard 4 lines of text at the top of the screen to indicate the machine status as well as a graphical depiction of the cut file.

Shift Mode is indicated by the arrow at the bottom right (i.e., Normal Mode , Shift Mode ).

The computer connection is also indicated at the bottom right (i.e., connected , not connected ). Text will appear at the bottom to indicate what action is occurring or needs to be accomplished before the cutting sequence can be started (e.g., **Machine Not Homed!**  ). Operators may press the

X-axis Down Jog Arrow  to view additional menu items in the various menus available on the keypad.

The keypad also provides direction with the color scheme: **Blue keys** indicate adjustments and settings; **Green keys** indicate action or movement; and the **Yellow key** indicates caution or pause. The Emergency Stop (E-Stop) **red button** located at the top of the keypad allows the operator to immediately shut down or discontinue the current job.



[Keypad](#)

Spindle Warmup

The Spindle Warmup is a procedure required by spindle manufacturers before applying load so that the bearings, bearing housings, and shaft are allowed to expand to their design dimensions. Applying a load to a cold spindle will load the bearing unevenly, causing bearing failure. Operators should insert a tool into the spindle prior to warmup. If the spindle is air or water cooled, then the operator should make sure that the cooling is turned off during spindle warmup.

After booting the machine and finding Home, the operator will receive the following prompt from the keypad:



Run Spindle Warmup
using Tool #? 1
Cancel for NO Warmup

The default tool is #1, so the operator can press **Enter**  to warm up the spindle with this tool. If another tool is preferred, the operator can press the number of that tool or press **Cancel**  to skip the spindle warmup process.

When **Enter**  is pressed, the machine picks up the tool and performs the spindle warmup procedure. This procedure will run at 6000 RPM for 5 minutes, 12000 RPM for 5 minutes, and then 18000 RPM for 5 minutes.

While the procedure runs, the system displays the remaining time for each RPM.

[Keypad](#)

Emergency Stop (E-Stop)

Pressing **Emergency Stop (E-Stop)** on either the handheld keypad or at the side of the gantry cuts power to everything except the control board of the unit, which will go into Emergency Stop mode and cut power to the inverter. This causes the spindle to coast to a stop, a process that may take several minutes. All job setup information is erased from the system, and operators should wait at least 2 minutes before engaging the machine and applying power to the motors. The **E-Stop** button must be pulled out when the main power switch is turned on.


[Keypad](#)

Hot Keys for the Router

Hot Keys are the keys on the **MultiCam** Handheld Keypad which access functions without using the menu system. The operator can access different aspects of the system using the Hot Keys, which are listed under Normal Hot Key Functions and Shifted Hot Key Functions.

Normal Hot Key Functions (as shown on keypad)

 Go Home	 Feedrate Increase	 Jog Z Up	 Spindle RPM Increase	 Pause
 Set Surface	 Feedrate Decrease	 Jog Z Down	 Spindle RPM Decrease	 Start
 Set Home	 Move (7)	 Jog Up (8)	 Spindle Test (9)	 Menu
 Set Max Depth	 Jog Left (4)	 5	 Jog Right (6)	 Shift
 DNC	 Bar Code (1)	 Jog Down (2)	 Drive Enable (3)	 Cancel (Exit)
 Jog Speed	 Help (+/-)	 0	 Soft Homes (.)	 Enter

Additional hot key features can be accessed when the keypad buttons are pressed in combination with

Shift 

Shifted Hot Key Functions

 Find Home		 Park Z Up	 Spindle RPM	
 Fasmer Surface		 Park Z Down	 Spindle Mode	 Proximity Restart
 Return to Home 0,0		 Park X High	 Mister Test	
 Adjust Max Depth	 Park Y High	 Open Chuck	 Park Y Low	
		 Park X Low	 Drive Disable	
	 Controller Info	 Execute Self Test	 Set Soft Homes	

[Keypad](#)

Go Home

The Go Home function is a return to the current 0,0 Soft Home and Tool Lift. The coordinates can be modified, but most operators prefer using soft homes for establishing different home coordinates during the cutting process.

Press Go Home .



[Hot Keys](#)

[Keypad](#)

Feedrate Override

The Feedrate Override function allows the operator to adjust the X,Y travel of the machine and to change speeds from as little as 10% to as much as 100%. This affects the feedrate or cut speed when the cutter is moving through the material. The Feedrate Override has no effect when the machine is executing a rapid move above the material.

The cut speed will be shown on the keypad display as a percentage of the actual speed: *Feed%= ###*. The Feedrate Override determines the percentage by which the operator adjusts the cut speed.

1. Press **Feedrate Override**  to adjust the feedrate in 1% increments when a file is being executed on the machine.
2. Hold **Feedrate Override**  to adjust the feedrate in faster moving 1% increments when a file is being executed on the machine. [Hot Keys](#) [Keypad](#)

Jog Z-axis

1. Press and hold **Z-axis Up Jog Arrow**  or **Z-axis Down Jog Arrow**  to raise or lower the spindle.
2. Press **Jog Speed**  to increase or decrease the Jogging speed of the spindle. [Hot Keys](#) [Keypad](#)

Spindle RPM Override

The Spindle RPM Override function allows the operator to control the RPM of the spindle from minimum to maximum levels. The spindle RPM can be adjusted while a file is being executed, and the current spindle RPM will be shown on the keypad display as *RPM=####*. The spindle speed will be set to the new value whenever the system receives a spindle speed command either automatically while a file is being executed or manually when the operator uses the Hot Keys.

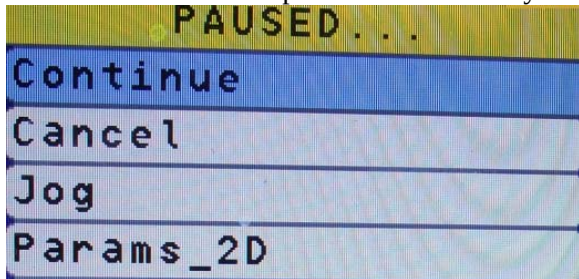
1. Press **Spindle RPM Increase**  or **Spindle RPM Decrease**  to adjust the spindle RPM by 100 RPM increments.
2. Hold **Spindle RPM Increase**  or **Spindle RPM Decrease**  to adjust the spindle RPM by faster moving 100 RPM increments. [Hot Keys](#) [Keypad](#)

Pause

The Pause function allows the operator to instantly stop any motion of the machine or any file. The Z-axis will move to the Tool Lift position, and the spindle will shut off. At this point, the operator can change any 2D cutting parameters or Jog the spindle out of the way to get a better look at the cut. When *Continue* is selected, the machine will automatically return to the position before Pause was activated and continue with the commanded motion or the file.



1. Press **Pause** to stop the machine for any reason.



2. Press **Jog Arrows** until the blinking cursor is over the first letter of the command to be executed.
 - a. **Continue** finishes out the operation or file originally executed.
 - b. **Cancel** ends the operation or file executed and shows *Motion Cancelled...* on the keypad display.
 - c. **Jog** allows the operator to Jog the machine to any location within the table parameters.
 - d. **Params_2D** allows the operator to make any quick changes in the 2D cut parameters.



3. Press **Enter** to accept the command.

[Hot Keys](#)

[Keypad](#)

Set Surface (Z=0)

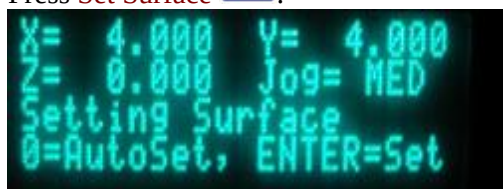
The Set Surface function allows the operator to identify and set the surface of the material or the software Z reference point as the Z-axis Home (Z=0). Once set, the surface will not have to be reset unless the system is shut off or the material thickness changes.

1. Make sure the vacuum pump is on. The vacuum pump will suck the material down so that the router can read the surface dimensions properly.

2. Press Jog Arrows  to position the tool bit anywhere over the material.

3. Place the grounding collar on the ATC mounting groove in the spindle tool holder.

4. Press **Set Surface** .



5. Place the surface block on top of the material underneath the tool.

6. Press **Z-axis Jog Arrows**  to lower the spindle until the bit is just above the surface block. The operator should monitor the Jog speed when lowering the Z-axis as the machine will only recognize the surface block when 0 is pressed. The spindle could crash into the material or the surface block and break the bit.

7. Press and hold **Zero**  to begin the surfacing routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height.

8. Remove the grounding collar and surface block before beginning work.

[Hot Keys](#)

[Keypad](#)

Start

Pressing **Start**  will begin a job file or execute a Cut Utility. When a file has been completed, the operator can press **Start**  to activate the replay buffer and begin the file again. Operators should never leave the machine unattended during the cutting sequence.

[Hot Keys](#)[Keypad](#)

Set Home (0,0)

The Set Home function allows the operator to identify the exact location of X=0, Y=0 on the material to be cut. This is the Soft Home location 0. The file that is to be executed on the table will use this location as the software reference point home or origin (0,0): the machine will begin and return to this new origin, and the X and Y coordinate locations are stored in the controller's memory. The location of the Set Home position will be the new origin (0,0) until a new home is set or the current home is cleared by selecting Clr_Home from the Utility sub-menu. The software reference point **must be** the material home point.

1. Press **Jog Arrows**  to place the **center of the bit** over the front right corner of the material.

2. Press **Set Home** .



3. Press **Enter**  to accept. This will store the absolute X and Y coordinate locations in the controller memory.

[Hot Keys](#)[Keypad](#)

Move

The Move function allows the operator to enter exact values for X, Y, Z (0,0,0) where the first entry is X (length), the second entry is Y (width), and the third entry is Z (height). The X, Y, and Z values will reference from the Hard Home position until the Set Home and Set Surface functions have been completed. At that point, the X- and Y-axes will reference from the Set Home position. The Z value will reference from the surface of the material.

1. Press **Move** .
2. Enter the desired value for X using the number keypad (e.g., 4.0) and press **Enter** .
- 
3. Enter the desired value for Y using the number keypad (e.g., 4.0) and press **Enter** .
- 
4. Enter the desired value for Z using the number keypad (e.g., 0.0) and press **Enter** .
- 
5. Press **Enter**  when the “?” is displayed to accept all settings and move the machine to the new coordinates.
- 
6. Press **Cancel**  to return to the main display.

The Move function can also be used to verify a specific location.

1. Press **Move** .

2. Press **Enter**  three times to verify the current location of the X-, Y-, and Z-axes.

The operator should press **Enter**  to keep the axis value from changing when that value is shown on the screen.

[Hot Keys](#)

[Keypad](#)

Jog X- and Y-axes

The **Jog Arrows**  are used to Jog the location of the spindle and to move through the menu.

Jog Speed  can be pressed to change the Jogging speed of the machine.

1. **Left Arrow**  is the Y-axis Positive Jog Key and Left Scroll Key in the menu.

2. **Up Arrow**  is the X-axis Positive Jog Key and Line Up Key in the menu.

3. **Down Arrow**  is the X-axis Negative Jog Key and Line Down Key in the menu.

4. **Right Arrow**  is the Y-axis Negative Jog Key and the Right Scroll Key in the menu.

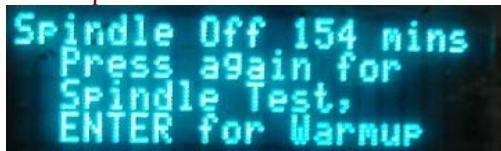
[Hot Keys](#)

[Keypad](#)

Spindle Test

The Spindle Test function tests the controller output to the spindle motor, which will stay active until the test key is pressed again or the test is cancelled.

1. Press **Spindle Test** .



2. Press **Spindle Test**  again to run the spindle test or press **Enter**  to run Spindle Warmup.

3. Press **Cancel**  to stop the Spindle Test.

[Hot Keys](#)
[Keypad](#)

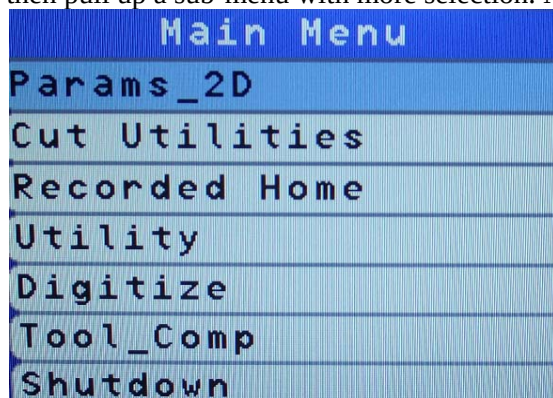
Menu

Menu items help the operator handle many of the functions of the **MultiCam** CNC Cutter by providing the ability to set feedrates and other parameters, run tests, and cut basic shapes. With the addition of some advanced features of the **MultiCam** Router, the operator can also adjust tools or duplicate shapes which create G-Code files quickly.

Many of the Menu Items are standard for all **MultiCam** Routers but with different options, such as Automatic Tool Change and MultiHead. Some options have further additional items.

1. Press **Menu**  to access the menu system.

2. Press **Jog Arrows**     to move the cursor to the specific menu item. Certain menu items will then pull up a sub-menu with more selection. Not all menu items will be applicable for some systems.


[Hot Keys](#)
[Keypad](#)

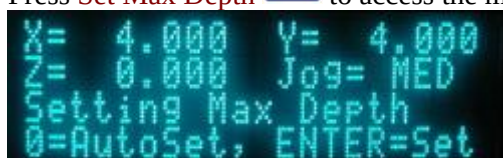
Set Max Depth

The Set Max Depth function allows the operator to set a cutting limit so that the tool will not plunge below that setting. The **MultiCam** Controller has a maximum depth safety feature that functions for all file types and modes and will truncate any depth greater than the maximum depth. Setting the maximum depth is similar to setting the surface, and the maximum depth also becomes the cut depth. The operator may set a shallower cut depth after the maximum depth has been configured.

1. Press **Jog Arrows**  to position the tool bit anywhere over the tabletop or at the location of the required maximum depth.

2. Place the grounding cradle on the ATC mounting groove in the spindle tool holder.

3. Press **Set Max Depth**  to access the maximum depth for the bit.



4. Place the surface block underneath the tool and on top of the tabletop or at the required maximum depth.

5. Press **Z-axis Jog Arrows**  to lower the spindle until the end of the tool is just above the surface block.

6. Press and hold **Zero**  to begin the Maximum Depth routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height.

7. Remove the grounding collar and surface block before beginning work.

[Hot Keys](#)
[Keypad](#)

Shift

Pressing **Shift**  before pressing any Hot Key will allow the operator to toggle between *Normal* Hot Key and *Shifted* Hot Key functions.

When **Shift**  has been pressed, the Shift LED will illuminate to indicate that the keypad is now in *Shifted Mode*.

[Hot Keys](#)
[Keypad](#)

DNC

The Distributed Numerical Control (DNC) function allows the operator to access files directly from the hard disk of the host PC by using the keypad at the machine. This allows the computer and the machine to be in different locations, which frees up the operator to view the progress of the cutting sequence.

1. Make sure that JobNameServer  has been activated on the computer before selecting this function.

2. Press **DNC** .



The keypad will display a list of the directories in the DNC path after the system has validated and verified.

3. Press **Jog Arrows**  until the blinking cursor is over the directory to be entered.

4. Press **Enter** . The keypad will display the DNC files that are available in that directory, and directories or folders and any associated subfolders are displayed between brackets []. Press **Enter**  while the blinking cursor is located over the 2 decimal points at the beginning of the directory to return to the list of directories if necessary.

5. Press **Jog Arrows**  until the blinking cursor is over the first letter of the file to be executed.

Z-axis Jog Arrows  will page up or down one screen at a time, while **X-axis Jog Arrows**  will move up or down one line at a time.

6. Press **Start** .



If the operator just presses **Enter**  over the blinking cursor, he or she will only receive a display of the file name, size, and date/time created.



7. Press one of the following Hot Keys to execute the required command or press **X-axis Down Jog Arrow** once to access the Dry Run function and once more to access the Step and Repeat function.

- a. **Start**  will execute the selected job file.
- b. **Cancel**  will end the selected job file.
- c. **Pause**  will allow the operator to change a tool, set up a 2D parameter, or set surface or max depth while holding the file in queue.

Once the file has completed running, the keypad display will show how long the selected file ran and will automatically return to the directory.

[Hot Keys](#)

[Keypad](#)

Bar Code

The Bar Code function allows the operator to scan a label with the Bar Code Scanner, which will automatically call up and download the correct job for execution on the machine. The Bar Code Scanner is limited to 48 characters excluding the initial DNC path set up in JobNameServer (e.g., C:\DNCfiles\).

- 1. Press **Bar Code**  to execute the Bar Code function. This will set the display to show when the system is bar code ready.
- 2. Depress the trigger on the bar code scanner and scan the bar code label.

Example Only:



- 3. Press **Start**  when prompted to begin the job.

[Hot Keys](#)

[Keypad](#)

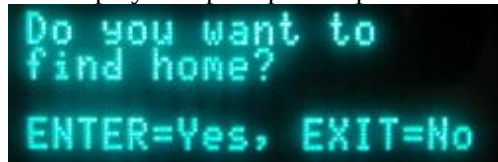
Drive Enable

1. Press **Drive Enable**  to enable all Servo Drives after a system error.



Motor Drives
Enabled...

The display will prompt the operator to find home after re-enabling the motor drives.



Do you want to
find home?
ENTER=Yes, EXIT=No

2. Press **Enter**  to ensure proper machine operation.

[Hot Keys](#)

[Keypad](#)

Cancel

Press **Cancel**  to cancel or exit any selected function or menu item. In some cases, this needs to be pressed twice to make sure the function has been completely canceled.

[Hot Keys](#)

[Keypad](#)

Jog Speed

Press **Jog Speed**  to toggle between 3 different speeds at which the machine can Jog the **MultiCam** Router. The keypad display will prompt the operator with *Jog= FAST, MED, or SLOW*.

[Hot Keys](#)

[Keypad](#)

Help

The Help function allows the operator to learn the functions of any Hot Key selected. There are 2 modes of the Help function: *Normal* and *Shifted*. In *Normal* mode, the operator can review the standard function of any Hot Key on the display. In *Shifted* mode, the operator can review the second or shifted function of that key on the display.

1. Press **Help**  to access the Hot Key Help Feature.
2. Press any key for a brief description of that key in *Normal* mode.
3. Press **Shift**  and any key for a brief description of that key in *Shifted* mode.
4. Press **Cancel**  twice to exit help.

[Hot Keys](#)[Keypad](#)

Soft Homes (.)

The Soft Homes function allows the operator to have multiple fixture setups to be quickly accessed during the cutting procedure. The Soft Homes Hot Key also allows the operator to input a decimal into a numeric value.

1. Press **Soft Homes** .



2. Enter the appropriate number (1-9) to assign the proper value for the new home position (e.g., 6).
3. Press **Enter**  to accept the Soft Home. The display will acknowledge that the particular home position has been selected.



4. Press **Shift**  and **Soft Homes**  to set new Soft Home locations.

[Hot Keys](#)[Keypad](#)

Enter

Press **Enter**  to accept functions, settings, and menu items as directed.

The display may provide a question mark “?” when processing certain functions. The operator can press

Enter  to verify that the action should take place.

[Hot Keys](#)

[Keypad](#)

Find Home

The Find Home function allows the operator to return each axis to the machine home position in order to relocate the homing targets. The Spindle Head Assembly will move to the right front of the table and index all X, Y, Z components starting with the Z-axis.

1. Press **Shift**  and **Go Home**  to index the gantry, carriage, and Z assembly off of the limit switches after one of the following actions have been completed:
 - a. The machine has just been turned on.
 - b. The machine has stalled or lost its position coordinates.
 - c. The machine has been stopped because the operator has pushed **E-Stop**.
2. Refer to the display as it will indicate when the system locates the targets for each axis. The spindle drive will be enabled when the Find Home function has been activated. The operator should take special care ATC systems in making sure the air is **on** and the dust collector is **open**.

[Hot Keys](#)

[Keypad](#)

Park Z Up or Down

The Park Z function will move the spindle head assembly to either the absolute Z=0 position or all the way up in a process which allows for tool changes including easily moving the pressure foot.

1. Press **Shift**  and **Z-axis Up Jog Arrow**  to raise the spindle head assembly to the absolute Z=0 position.
2. Press **Shift**  and **Z-axis Down Jog Arrow**  to lower the spindle head assembly to the Tool Lift position.

[Hot Keys](#)

[Keypad](#)

Spindle RPM

The Spindle RPM refers to the revolutions per minute that the spindle will spin, and the spindle RPMs may vary depending on the type of material being cut. Operators can enter the desired spindle RPM for manual operations such as Material Cutoff or Table Mill. The Spindle RPM can also be set in the cut file and will override what has been entered at the keypad. This value can be set from the keypad or in the Params_2D sub-menu. The keypad display is shown below.

1. Press **Shift**  and **Spindle RPM Increase** .

2. Enter the desired spindle RPM value using the number keypad (e.g., 18000).



3. Press **Enter**  to accept the value. The display will show that the speed has been set at the value entered and will then default to the main screen.

[Hot Keys](#)

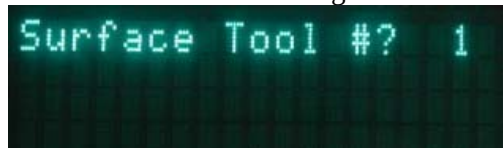
[Keypad](#)

Fasmer Surface

The Fasmer Surface function is useful when working with material that has an irregular surface.

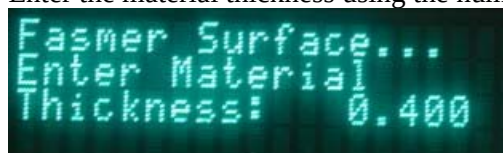
1. Press **Shift**  and **Set Surface**  to access the Fasmer Surface function.

2. Enter the Tool number using the number keypad (e.g., 1) and press **Enter** .



Surface Tool #? 1

3. Enter the material thickness using the number keypad (e.g., 0.400) and press **Enter** .

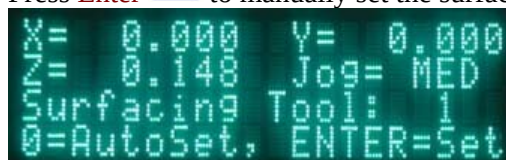


Fasmer Surface...
Enter Material
Thickness: 0.400

4. Place the surface block on the spoilboard.

5. Press **Jog Arrows**  to position the tool bit over the surface block.

6. Press **Enter**  to manually set the surface.



X= 0.000 Y= 0.000
Z= 0.148 Jog= MED
Surfacing Tool: 1
0=AutoSet, ENTER=Set

7. Press **Z-axis Jog Arrows**  to lower the spindle until the bit is just above the surface block. The operator should monitor the Jog speed when lowering the Z-axis as the machine will only recognize the surface block when **Zero** is pressed. The spindle could crash into the material or the surface block and break the bit.

8. Press and hold **Zero**  to begin the surfacing routine. This will cause the Z-axis to move down slowly until the tool comes in contact with the surface block and then move up above the surface block to the Tool Lift Height.

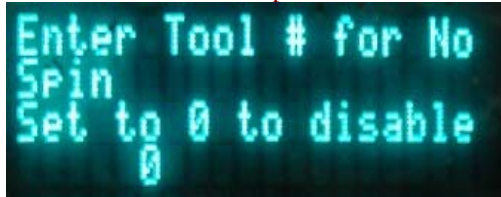
[Hot Keys](#)

[Keypad](#)

Spindle Mode

The Spindle Mode function allows the operator to enable or disable a tool. Changing a tool can be accomplished without the spindle inadvertently being turned on. Once the spindle has been disabled, the operator cannot activate the spindle from the keypad or a job file.

1. Press **Shift**  and **Spindle RPM Decrease**  to enter Spindle Mode.



2. Enter the Tool # for No Spin using the number keypad. The operator may enter **Zero**  to disable the function.
3. Press **Enter**  to accept the selection and return to the main display. [Hot Keys](#) [Keypad](#)

Proximity Restart

The Proximity (Prox) Restart function is an advanced feature that allows the operator to begin a file by locating the end position based on the original Home location. The controller will search through the file to locate the nearest point and base the coordinates of the restart on the Home position from where the file was originally executed.

Proximity Restart is used any time a file is interrupted and restarting the entire file is not practical. The operator can return to the original Home position to restart the program as the configuration is based on where the cutter stopped in relation to where the cutter originally began. This command will function through the internal replay buffer of the machine. The following directions describe how to operate Prox Restart for older systems:

1. Press **Shift**  and **Start**  from the Home position where the file was originally executed.



2. Press **Jog Arrows**     to move the spindle to a position within the restart tolerance of the desired point for restart.



3. Press **Enter**  to begin the search.

The keypad display will indicate the coordinates when a point within the restart tolerance is found. If no points are located within the restart tolerance, the system will default to the main display. At this point, the operator should repeat the Proximity Restart process.

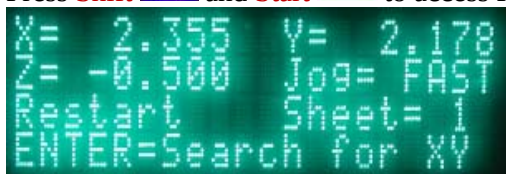
4. Press **Start**  to restart the file at the current location or press the **X-axis Up Jog Arrow**  to access the closest point in the file. Spindles can begin at the very beginning and retrace steps.
 - a. If the point is located over a Tool UP Move, follow the keypad display and repeat step 3.
 - b. If there is a power failure, the operator will need to re-index the machine and reset Surface and Max Depth.

For systems configured with Suite4.1 and Router Inits v.8.11.14, Prox Restart has been further modified. The following directions describe how to access the new Prox Restart function, select a particular mode, and complete basic troubleshooting.

Accessing Prox Restart

1. Jog to the appropriate location to begin the Prox Restart.

2. Press **Shift**  and **Start**  to access Prox Restart.



X= 2.355 Y= 2.178
Z= -0.500 Jog= FAST
Restart Sheet= 1
ENTER=Search for XY

- If the job is a single sheet, then skip to Step #5.
- If the job is not a single sheet, then continue with Step #3.

3. Press **Spindle RPM Override**   to select the sheet.



X= 13.037 Y= 3.914
Z= -18.104 Jog= FAST
Restart Sheet= 2
ENTER=Search for XY

4. Press **Start**  at the beginning of the selected sheet.

5. Press **Enter**  to search for the X,Y location. The machine will look for a point within a radius of 0.300" (7.620mm).
 - If the machine cannot find the point, then skip to **Troubleshooting Prox Restart**.
 - If the machine can find the point, then select a mode in which to begin the cutting sequence. Operators may access the different modes through either specific hot keys or keypad selection (shown below).

Operating in Hot Key Mode

The following hot keys have the same functionality as the same hot keys in Suite4 JobConsole. The system looks for a point closest to the current location based on the hot key pressed, and these hot keys operate independently of the mode currently visible on the display.

- The **X-axis Jog Arrows**   allow the technician to locate the previous or following point (e.g., perform the next Distance command: X-axis Up = Forward Distance, X-axis Down = Backward Distance).
- The **Z-axis Up and Down Jog Arrows**   allow the technician to locate the previous or next PD (i.e., Z-axis Up = Previous PD, Z-axis Down = Next PD).
- The **Feedrate Override Arrows**   allow the technician to locate where the previous or next tool will begin cutting (i.e., Feedrate Override Up = Previous Tool, Feedrate Override Down = Next Tool).

Operating in Keypad Selection Mode

Additional modes can also be accessed, which can be beneficial for operators who cannot find the starting point in the file through the above modes. Operators may select a Proximity Restart Mode through the keypad by completing the following:

1. Press **Help**  and the **Y-axis Jog Arrows**   to enter the Prox Restart keypad selection mode.
2. Press **Help**  to cycle through the Prox Restart modes and **Y-axis Jog Arrows**   to navigate forward and backward within the selected mode. Each mode is described below.
 - Dist – Move forward and backward based on a distance. When in Distance Mode, technicians can press **Jog Speed**  to cycle through 3 distances (i.e., 0.010", 0.050", 0.100" [1mm, 2mm, 3mm]).
 - PD – Move to the previous and next pen down (PD) commands in the job file (e.g., M11, M12)
 - PU – Move to the previous and next pen up (PU) commands in the job file (e.g., M21, M22)
 - Tool – Move to the previous and next tools in the job file
 - Mcmd – Move to the previous and next source line or Motion Command in the job file

Troubleshooting Prox Restart

An error message of *Location Not Found* will appear if JobConsole cannot find a matching Marker Type based on the mode selected above. When this happens, the system will automatically re-enter Prox Restart at the current location, and the technician will need to select a different mode to continue the Prox Restart.

For example, if a technician uses Mode PD to move to the next PD in the job file and then switches back to Mode Dist, he or she cannot back up in Dist mode. Since JobConsole is at the beginning of the contour, it will not be able to back up and therefore will report the error message *Location Not Found*.

[Hot Keys](#)[Keypad](#)

Return to Home 0

The Return to Home 0 function will return the spindle to the last recorded set home position or Soft Home location 0. If power has been lost or the machine has been rebooted, Home 0 will become the new Hard Home.

Press **Shift**  and **Set Home** .



When the Home position has been reached:

[Hot Keys](#)[Keypad](#)

Park X High or Low

The Park X function will move the spindle and gantry assembly to a set park position that could be as far away as the maximum length of the table or all the way back in X-axis. This allows the operator to reload material on the table or examine work without interference from the spindle.

1. Press **Shift**  and **X-axis Up Jog Arrow**  to move the spindle and gantry assembly to the high or farthest set park position.



2. Press **Shift**  and **X-axis Down Jog Arrow**  to move the spindle and gantry assembly to the low or closest set park position.

[Hot Keys](#)[Keypad](#)

Mister Test

The Mister Test function will test the controller output to the Automatic Misting System. On systems with no Automatic Misting System, this command will do nothing. This option can also be used to prime a misting unit if applicable.

1. Press **Shift**  and **Spindle Test**  to activate the Mister Test.



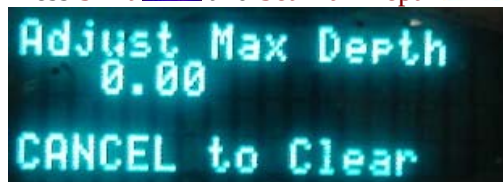
2. Press **Cancel**  to stop the Mister Test and return to the main display.

[Hot Keys](#)[Keypad](#)

Adjust Maximum Depth

The Adjust Maximum Depth function may be necessary when the cutter travels shallower than expected, usually due to the variance in material thickness. Operators can adjust the maximum depth without having to reconfigure all settings.

1. Press **Shift**  and **Set Max Depth** .



2. Press **Cancel**  to clear the maximum depth without having to reboot the machine or press

Z-axis Jog Arrows   to adjust maximum depth in one-thousandth increments (e.g., 0.001). A positive value will increase the range of maximum depth, and a negative value will decrease the range of maximum depth.

3. Press **Enter**  to accept the setting.

[Hot Keys](#)[Keypad](#)

Park Y High or Low

The Park Y function will move the spindle assembly the maximum length of the gantry to the right or left in Y-axis. This allows the operator to examine the work without interference from the spindle.

1. Press **Shift**  and **Y-axis Left Jog Arrow**  to move the spindle assembly to the left side of the gantry.



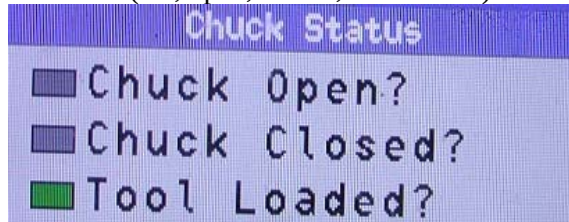
2. Press **Shift**  and **Y-axis Right Jog Arrow**  to move the spindle assembly to the right side of the gantry.

[Hot Keys](#)[Keypad](#)

Open Chuck

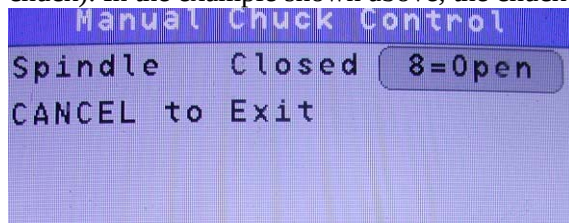
The Open Chuck function allows operators to open or close the chuck and load a tool. Once tools have been removed or modified in the ATC, the chuck must be closed before work can be continued.

1. Press **Shift**  and **Five**  to access the manual chuck control. The display will show the current status of the chuck (i.e., open, closed, tool loaded).

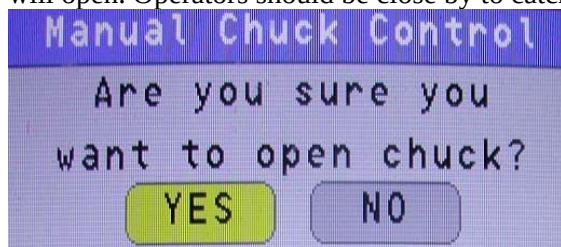


Operators may press **Cancel**  to exit the Open Chuck screen.

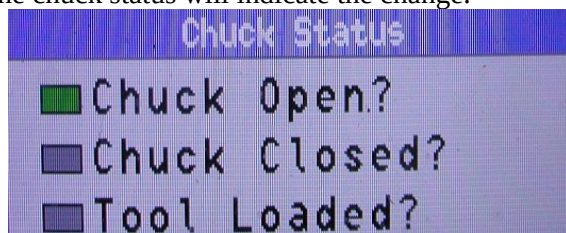
2. Press the number key indicated on the keypad (i.e., **Eight**  to open the chuck, **Two**  to close the chuck). In the example shown above, the chuck must be opened to unload the tool.



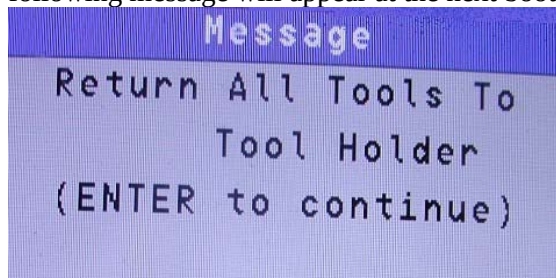
3. Press **Y-axis Jog Arrows**   until the option is highlighted (e.g., Yes) and press **Enter** . The chuck will open. Operators should be close by to catch the tool before it falls from the open chuck.



The chuck status will indicate the change.

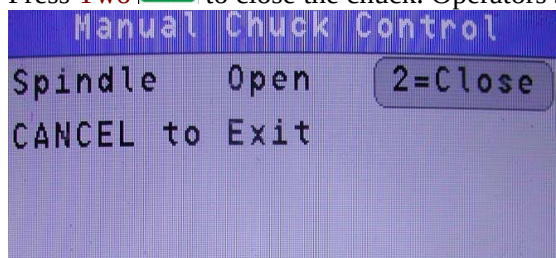


4. Load a tool if needed. Operators should remove tools before powering down their machines; otherwise, the following message will appear at the next boot up.

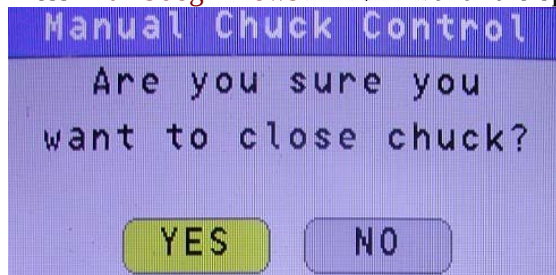


If this message appears at bootup, operators should start at step #1 and complete the entire process.

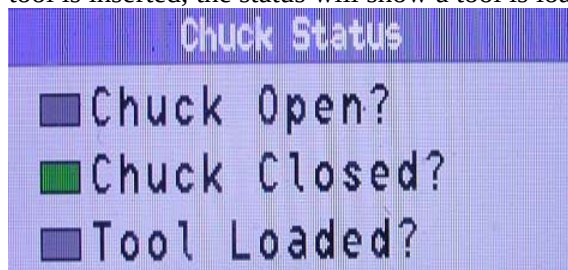
5. Press **Two**  to close the chuck. Operators should close the chuck even if there is no tool inserted.



6. Press **Y-axis Jog Arrows**   until the option is highlighted (e.g., Yes) and press **Enter** .



The chuck status will indicate the change. If no tool is inserted, the status will show the chuck is closed. If a tool is inserted, the status will show a tool is loaded.



[Hot Keys](#)

[Keypad](#)

Drive Disable

The X-, Y-, and Z-axis drives will be disabled in the event of a system failure or a mechanical obstruction to prevent possible damage to the individual drives. The operator also has the ability to disable the drives from the keypad. Once the drives have been disabled, the machine cannot be operated by using the keypad.

Press **Shift**  and **Drive Enable**  to disable all drives.



When the drives associated with servo systems are disabled, the power to the motors is also disabled. The motors associated with stepper systems still have power after the drives are disabled, but there is no output from the control board.

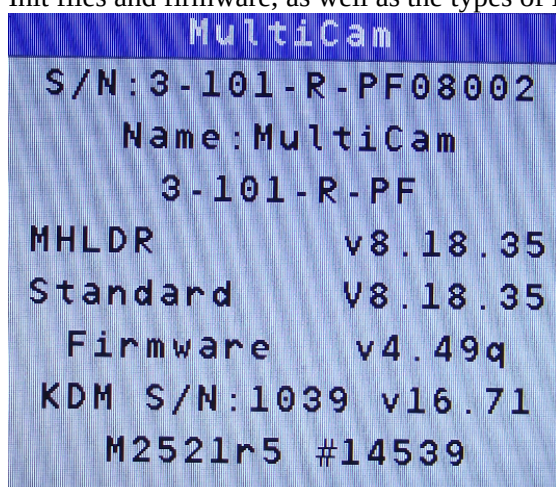
[Hot Keys](#)

[Keypad](#)

Controller Information

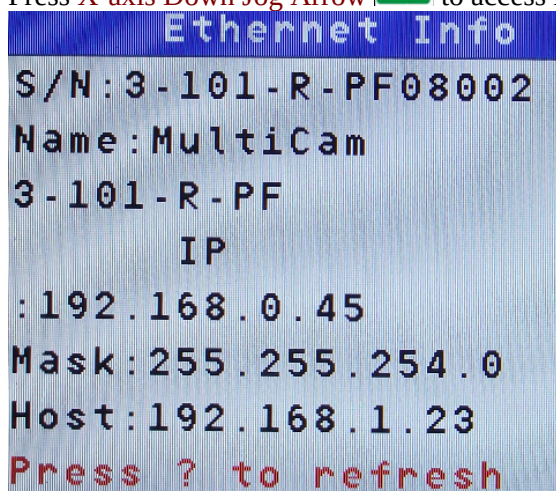
The Controller Information function provides information on the versions of software that are loaded in the controller.

1. Press **Shift**  and **Help**  to display the controller information. This includes the version of machine Init files and firmware, as well as the types of Interface and Control boards.



```
MultiCam
S/N:3-101-R-PF08002
Name:MultiCam
3-101-R-PF
MHLDR v8.18.35
Standard V8.18.35
Firmware v4.49q
KDM S/N:1039 v16.71
M2521r5 #14539
```

2. Press **X-axis Down Jog Arrow**  to access Ethernet information while viewing the controller information.



```
Ethernet Info
S/N:3-101-R-PF08002
Name:MultiCam
3-101-R-PF
IP
:192.168.0.45
Mask:255.255.254.0
Host:192.168.1.23
Press ? to refresh
```

3. Press **DNC**  to change the IP address if necessary.

[Hot Keys](#)[Keypad](#)

Execute Self Test

Press **Shift**  and **Zero**  to access the self test, which runs a diagnostic on the following:

- Memory verification
- Rectangle routing
- Surface block checking
- Automatic Tool Changer (ATC) testing
- Origin (Home) testing

[Hot Keys](#)[Keypad](#)

Set Soft Homes

The Set Soft Homes function allows the operator to establish 9 programmable Soft Home positions to record multiple home positions that can be used in supporting multiple cutting fixtures. These homes stay recorded in memory even after power has been removed from the system.

1. Press **Jog Arrows**  to position the **center of the bit** where the new Soft Home location should be set.

2. Press **Jog Speed**  to change the Jogging speed if necessary.

3. Press **Shift**  and **Soft Home** .



4. Enter a number between 1 and 9 using the number keypad. If the operator wants to save the Z value in the Soft Home, he or she should contact a **MultiCam** technician for assistance.

5. Press **Enter**  to record the new Soft Home.

[Hot Keys](#)[Keypad](#)

Advanced

Advanced Hot Keys are accessed by pressing **Shift** , **Help** , and any of the keypad numbers. The operator will begin in the main information screen.



Main Information Screen

```
MultiCam
S/N: 3-101-R-PF08002
Name: MultiCam
3-101-R-PF
MHLDR v8.18.35
Standard V8.18.35
Firmware v4.49q
KDM S/N: 1039 v16.71
M2521r5 #14539
```

“MultiCam”

Serial Number of the Machine

Machine Name

Machine Size

Init File Version

Firmware Version

Keypad Type and Serial Number

Board Type and Serial Number



Ethernet Information Screen

```
Ethernet Info
S/N: 3-101-R-PF08002
Name: MultiCam
3-101-R-PF
IP
: 192.168.0.45
Mask: 255.255.254.0
Host: 192.168.1.23
Press ? to refresh
```

“MultiCam”

Serial Number of the Machine

Machine Name

Machine Size

IP Address

Net Mask

DHCP Server Address (only shows if
DHCP enabled)

“Press ? to refresh”

Press **DNC**  from this screen to be prompted to enable DHCP or specify an IP address.



Power Information Screen

```
PS3000 v908
Web Voltage 24.00V
Servo Power 78.42V
Stepper Power 61.71V
24V Current 1.08A
Run HvOK Fan Fault
```

Power System and Version

Web Voltage

Servo Power

Stepper Power

24V Current

Power Status



MODBUS Information Screen

```

MODBUS Information
Sent:1730
TMOs:4
Blocked:0
--MB Loopback Test--
FPGA ... Pass
HEXT ... Pass
HINT ... Pass
TH1 ... Pass
    
```

“MODBUS Information”

Packets Sent

Timeouts

Blocked Responses

“MB Loopback Test”

FPGA

HEXT

HINT

TH1



PCB Temperature

```

P C B   T e m p e r a t u r e

1 1 4 . 8 4   D e g F
    
```

“PCB Temperature”

Temperature of the PCB



Inputs Information Screen

```

Info Inputs
Xa  Xb  Ya  In #4
Z1  In #5 In #7 In #8
Pause Cyl#1 Cyl#2 In#12
2DDig CE  In#15 In#16
Surf GClip Inv#1 Inv#2
    
```

Each box shows the status of the specific input.

- Dark = off
- Bright = on



Machine Information Screen

```

Machine Information
Model: MultiCam 3-10
S/N   : 3-101-R-PF080
02
    
```

“Machine Information”

Model Name

Serial Number

[Keypad](#)

Advanced Features

Advanced Features help the operator to be more productive by previewing a file before cutting the material with [Dry Run](#), rotating a file to match the angle of the sheet with [Rotate File](#), and cutting an entire sheet of shapes using the [Step and Repeat](#) function. Some routers are also available with [Dual Start](#), an option to maximize the cutting time by creating 2 separate work areas on the same machine.

[Keypad](#)

Dry Run

The Dry Run function is an advanced feature that allows the operator to preview the file to be cut while leaving the feedrate override active. The spindle will stay in the Tool UP position, never entering the material. The keypad displays *Tool 1 Up* when the cutter would normally be in the UP position moving between contours. The keypad displays *Tool 1 Down* when the cutter would normally be in the material cutting the contours. This command will function through the DNC or through the internal replay buffer of the machine.

1. Make sure that JobNameServer  has been activated on the computer.

2. Press **DNC** . The keypad display will show a list of the directories in the DNC path.

3. Press **Jog Arrows**  until the blinking cursor is over the directory to be entered.

4. Press **Enter** . The keypad will display the DNC files that are available in that directory, and directories or folders and any associated subfolders are displayed between brackets []. Press **Enter**  while the blinking cursor is located over the 2 decimal points at the beginning of the directory to return to the list of directories if necessary.

5. Press **Jog Arrows**  until the blinking cursor is over the first letter of the file to be executed.

6. Press **Start** .

```
TonyCircles.nc(1)
*START to Execute
*CANCEL to Abort
*PAUSE to Yield
```

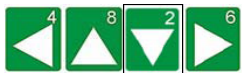
7. Press **X-axis Down Jog Arrow**  once.



8. Press **Enter**  to start the Dry Run process. The operator can press **Cancel**  to abort the selected job file or **Pause**  to change a tool, set up a 2D parameter, or set surface or max depth while holding the file in queue. [Advanced](#) [Keypad](#)

Rotate File

The Rotate File function is used to rotate the program to match the material. Operators may find it impractical to square material sufficiently when heavy materials are loaded onto the table, so this function allows the machine to adjust the parameters of the job file.

1. Press **Jog Arrows**  to position the **center of the bit** over the rear right corner of the material and press **Set Home** .



2. Press the **X-axis Down Jog Arrow**  to set the Rotation Point. The keypad will briefly display the rotation setting and then switch back to the main display.



3. Press **Start**  to begin the job file. The operator will be prompted to accept or reject the rotation. [Advanced](#) [Keypad](#)

Step and Repeat

The Step and Repeat function is used to make a single job file cut multiple times. Operators can program a single part of the cutting sequence and include the size of the material to be cut without having to create a larger file with the detailed parts and motions.

1. Make sure that JobNameServer  has been activated on the computer.

2. Access the file by pressing **DNC** .

3. Press **Jog Arrows**  until the blinking cursor is over the directory to be entered.

4. Press **Enter** . The keypad will display the DNC files that are available in that directory, and directories or folders and any associated subfolders are displayed between brackets []. Press **Enter**  while the blinking cursor is located over the 2 decimal points at the beginning of the directory to return to the list of directories if necessary.

5. Press **Jog Arrows**  until the blinking cursor is over the first letter of the file to be executed.

Z-axis Jog Arrows  will page up or down one screen at a time, while **X-axis Jog Arrows**  will move up or down one line at a time.

6. Press **Start**  to access the particular file.

```
TonyCircles.anc(1)
♦START to Execute
♦CANCEL to Abort
♦PAUSE to Yield
```

If the operator just presses **Enter**  over the blinking cursor, he or she will only receive a display of the file name, size, and date/time created.

7. Press **X-axis Down Jog Arrow**  twice to access the Step and Repeat function.

```
TonyCircles.anc(1)
Step And Repeat
♦ENTER to select
```

8. Press **Enter**  to start the Step and Repeat function.

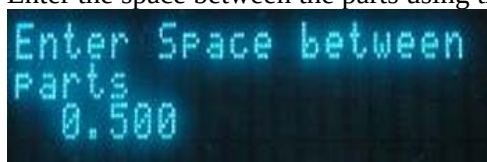
9. Enter the sheet size in X-axis using the number keypad (e.g., 119.565) and press **Enter** .



10. Enter the sheet size in Y-axis using the number keypad (e.g., 48.690) and press **Enter** .



11. Enter the space between the parts using the number keypad (e.g., 0.500) and press **Enter** .



12. Press **X-axis Jog Arrows**   to select whether the machine should step along the X- or Y-axis.



13. Press **X-axis Jog Arrows**   to select Yes or No for the Serpentine option.



If X-axis was selected above and serpentine is selected next, then the machine will cut the parts in a line down the X-axis, move over in the Y-axis, and cut back up the X-axis. This sequence of motion increases efficiency since the machine will not make a big slew move back to the front of the machine after each row of parts.

14. Press **Enter**  to start the cutting sequence.

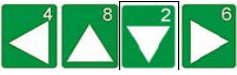
[Advanced](#)

[Keypad](#)

Dual Start

The Dual Start function is available with most CNC systems and requires an Operating (op) station with options for Home 1 and Home 2. Operators who run the same job several times in a row may choose the Dual Start option to maximize the table space and increase productivity.

1. Make sure the Dual Start Module has been loaded on the machine.
2. Set Parameter 24 to 1.
3. Set Soft Homes **1 and 2**. The system will automatically recognize that Dual Start is being requested. Other Home values will not work.

- a. Press **Jog Arrows**  to position the **center of the bit** where the first Soft Home location should be set.

- b. Press **Shift**  and **Soft Homes** .



- c. Enter **1**  for Home 1 and press **Enter**  to record the Soft Home.

- d. Press **Jog Arrows**  to position the **center of the bit** where the second Soft Home location should be set.

- e. Press **Shift**  and **Soft Homes** .



- f. Enter **2**  for Home 2 and press **Enter**  to record the second Soft Home.

4. Press **DNC** .

5. Press **X-axis Jog Arrows**  until the blinking cursor is over the appropriate job file.

6. Press **Enter**  to access the job file.
7. Push button 1 or 2 on the op station to move the spindle to Soft Home 1 or 2.
8. Press **Start**  to begin the file.
9. Press the other button on the op station to run the same job at the other Soft Home location after the first job file has been completed. Operators should make sure the material has been properly loaded at the other location before beginning the job file at that location. [Advanced](#) [Keypad](#)

Menu Items

[top](#)

Params_2D	4
X,Y Feedrate	5
Cut Depth	6
Tool Lift	7
Z Plunge Feed	8
Spindle RPM	9
Spindle DIR	10
Recorded Home	11
Digitize	12
Start Digitize	13
Auto Scan Rate	16
Auto Step Size	17
Scan Mode	18
Add Corners	19
Output Start	20
Shutdown	21
Cut Utilities	22
Table Mill	23
Table Mill Tool Parameters	25
Material Cutoff	26
Test Cut	27
Test Cut Tool Parameters	28
RipCut	29
Circle	30
Circle Shape Tool Parameters	31
Polygon	32
Polygon Shape Tool Parameters	33
Rectangle	34
Rectangle Shape Tool Parameters	35
Radiused Rectangle	36
Radiused Rectangle Shape Tool Parameters	38
Utility	39
Clear Home	40
Show Last Job Time	40
Set X Park Position	41
Tool Configuration	42
Adjust Surface	43
DNC Mode	44
Pause Configuration	46
Reset Inverter	47
Reboot	48
Lube	49
Dust Collector	50
Default Max Depth	51
MisterOpt	52

<u>No Spin</u>	53
<u>SPA Trigg</u>	54
<u>GDrillTest</u>	55
<u>Dust Collector Test</u>	56
<u>Tool Comp</u>	57
<u>Vision</u>	59
<u>Focus Camera</u>	60
<u>Fiducial Diameter</u>	61
<u>Correlation</u>	62
<u>Retake Distance</u>	64
<u>Settling Time</u>	65
<u>Flip to Cut</u>	66
<u>Auto Focus</u>	68
<u>Fast Find</u>	69
<u>Camera Delay</u>	71
<u>Auto Find Setting</u>	72
<u>Manual Find Setting</u>	73
<u>Set Camera Scale</u>	74
<u>Set Camera Offsets</u>	75
<u>Move to Cam Offsets</u>	78
<u>ATC</u>	79
<u>Calibrate Tool</u>	80
<u>Adjust Tool</u>	81
<u>Tool Unload Option</u>	82
<u>Get Tool</u>	83
<u>Load Tools</u>	84
<u>Calibrate Gang Drill</u>	85
<u>Load Drill</u>	86
<u>AdjGDrill</u>	87
<u>CfgGDrill</u>	88
<u>Knife</u>	89
<u>Set Home w/Knife</u>	90
<u>Knife Control</u>	91
<u>XY Feedrate</u>	92
<u>ZDn Feedrate</u>	93
<u>Knife Lift</u>	94
<u>Knife Depth</u>	95
<u>Closeness</u>	96
<u>Rotate Degree</u>	97
<u>Knife Dn Delay</u>	98
<u>Knife Up Delay</u>	99
<u>Min. Radius</u>	100

<u>KnifeUtil</u>	101
<u>Cartridge Number</u>	102
<u>Home Knife</u>	103
<u>Cartridge Config</u>	104
<u>Cartridge Offsets</u>	106
<u>Osc Stroke</u>	107
<u>Move to Knife Offset</u>	108
<u>Knife Osc Test</u>	109
<u>Set Knife 1 Res</u>	110
<u>Set Knife 2 Res</u>	111
<u>MatlHand</u>	112
<u>Roller Setup ...</u>	113
<u>Material Pusher ...</u>	114
<u>Material Lifter ...</u>	115
<u>Air/Knife Sweeper</u>	116
<u>PopUp Pins</u>	117
<u>Auto/All</u>	118
<u>Z Sync</u>	119

Params_2D

The Params_2D sub-menu is used to set machine parameters specific to a routing machine. The sub-menu items allow the operator to set certain specifications for cutting different materials or removing waste material. The settings configured in Params_2D are ignored with the executed CNC files.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until Params_2D is highlighted in blue.
3. Press **Enter**  to access the Params_2D sub-menu.
4. Press **X-axis Jog Arrows**  until the appropriate option is highlighted in blue. Certain menu items will then enter a sub-menu with more selections.

Params_2D	
X,Y Feedrate	: 600.0
Cut Depth	: 0.000
Tool Lift	: 0.500
Z Plunge Feed	: 60.00
Spindle RPM	: 18000
Spindle DIR	: FWD

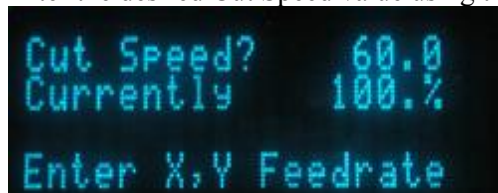
[X,Y Feedrate](#)[Cut Depth](#)[Tool Lift](#)[Z Plunge Feed](#)[Spindle RPM](#)[Spindle DIR](#)[Menu](#)

X,Y Feedrate (Cut Speed)

The X,Y Feedrate refers to the rate, measured in inches per minute (IPM), that the tool will travel in the X, Y direction through the material while the tool is in the Pen Down (PD) position (i.e., the Cut Speed). These Speed settings will vary with the type of material used. Metal is usually cut at a slower speed to prevent possible tool breakage, while plastic and wood generally have a faster cut speed to prevent material melting or burning.

The Cut Speed function can only be used when no feedrate has been programmed in the numerically controlled (NC) file. The machine will default to 60 IPM if the machine loses power or the controller is rebooted.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until Params_2D is highlighted in blue.
3. Press **Enter**  to access the Params_2D sub-menu.
4. Press **X-axis Jog Arrows**  until X, Y Feedrate is highlighted in blue.
5. Press **Enter**  to adjust the Cut Speed value.
6. Enter the desired Cut Speed value using the number keypad (e.g., 60).



The value shown next to “Currently” shows at what percentage the feedrate is set.

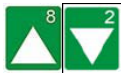
7. Press **Enter**  to register the value and return to the Params_2D sub-menu.

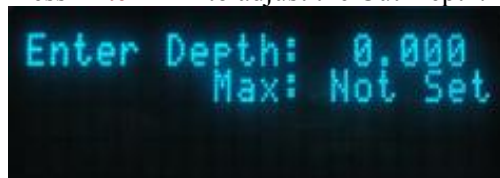
[Params_2D](#)

[Menu](#)

Cut Depth

The Cut Depth function is used to set how deep the cutter will go below the surface on a cutting move. The Cut Depth function at the controller is only applicable for 2 dimensional HPGL files that do not have the ZD command in them, and Cut Depth has **no effect** on G-Code files or 3D HPGL files.

1. Press **Menu**  to access the menu system.
 2. Press **X-axis Jog Arrows**  until Params_2D is highlighted in blue.
 3. Press **Enter**  to access the Params_2D sub-menu.
 4. Press **X-axis Jog Arrows**  until Cut Depth is highlighted in blue.
 5. Press **Enter**  to adjust the Cut Depth.
6. Enter the desired Cut Depth value using the number keypad.
7. Press **Enter**  to record the value and return to the Params_2D sub-menu.



Enter Depth: 0.000
Max: Not Set



Depth Set.

[Params_2D](#)[Menu](#)

Tool Lift

The Tool Lift function is used to set the distance the tool bit will rise above the surface of the material between cuts and is only functional for controller functions such as Go Home and Material Cutoff, HPGL programs with no Tool Lift included, and G-Code programs. This lift allows the spindle to travel above and across the material without hitting the hold-down clamps. Tool Lift can be modified after the surface is set. The Go Home command will move the Z-axis to the new tool lift position.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until Params_2D is highlighted in blue.
3. Press **Enter**  to access the Params_2D sub-menu.
4. Press **X-axis Jog Arrows**  until Tool Lift is highlighted in blue.
5. Press **Enter**  to adjust the Tool Lift.
6. Enter the desired Tool Lift value using the number keypad (e.g., 0.50).



7. Press **Enter**  to record the Tool Lift value. The number entered should be a positive number, but the display will show it as a negative number since movement up from the surface is considered negative while movement down from the surface is considered positive.
8. Press **Cancel**  to return to the Params_2D sub-menu.

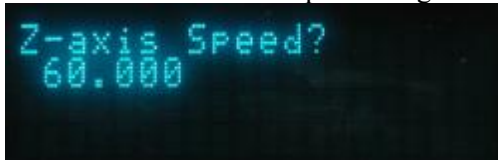
[Params_2D](#)[Menu](#)

Z Plunge Feed

The Z Plunge Feed is the feedrate in inches per minute (IPM) that the tool will travel in the Z-axis direction from surface to final depth, or the speed the tool enters the material. This speed is only active for 2-dimensional HPGL files that do not have a Z Speed or Cut Speed in them. Speed settings for hard materials should be slow, while speed settings for softer materials, such as plastic and wood, should be faster.

The machine will default to 60 IPM if the machine loses power or the controller is rebooted. The Z Plunge Feed does not control the speed by which the Z-axis travels from the Tool Lift position to the surface of the material. This speed is set in the machine parameters by **MultiCam** to reduce total process time.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until Params_2D is highlighted in blue.
3. Press **Enter**  to access the Params_2D sub-menu.
4. Press **X-axis Jog Arrows**  until Z Plunge Feed is highlighted in blue.
5. Press **Enter**  to adjust the Z Plunge Feed.
6. Enter the desired Z-axis speed using the number keypad (e.g., 60).



7. Press **Enter**  to record the value and return to the Params_2D sub-menu.

[Params_2D](#)[Menu](#)

Spindle RPM (Menu)

The Spindle RPM function is the revolutions per minute that the spindle will spin, and spindle speeds may vary with the type of material being cut. The operator can adjust the spindle speed with the Spindle RPM Override Hot Keys. The spindle speed can also be set in the cut file and will overwrite what has been entered in the keypad.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**   until Params_2D is highlighted in blue.
3. Press **Enter**  to access the Params_2D sub-menu.
4. Press **X-axis Jog Arrows**   until Spindle RPM is highlighted in blue.
5. Press **Enter**  to access the Spindle RPM function.
6. Enter the desired Spindle RPM value using the number keypad (e.g., 18000).



7. Press **Enter**  to register the value and return to the Params_2D sub-menu.

[Params_2D](#)[Menu](#)

Spindle DIR

The Spindle DIR function is used to adjust the direction the spindle will turn.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**   until Params_2D is highlighted in blue.
3. Press **Enter**  to access the Params_2D sub-menu.
4. Press **X-axis Jog Arrows**   until Spindle DIR is highlighted in blue.
5. Press **Enter**  to toggle between Forward (FWD) and Reverse (REV).



[Params_2D](#)

[Menu](#)

Recording a Soft Home

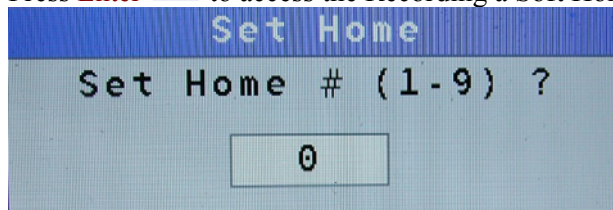
There are 9 different recordable home positions that can be used to support multiple cutting fixtures. These homes stay recorded even after power has been removed from the system.

1. Press **Jog Arrows**  to position the **center of the bit** where the new Soft Home location is to be set.

2. Press **Menu**  to access the menu system.

3. Press **X-axis Jog Arrows**  until Recorded Home is highlighted in blue.

4. Press **Enter**  to access the Recording a Soft Home function.



5. Enter the desired number (1-9) to record the new home.

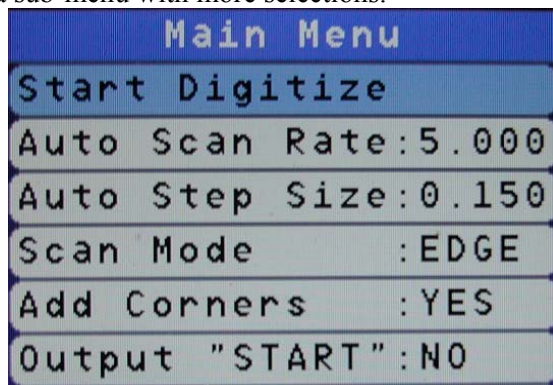
6. Press **Enter**  to return to the main menu.

[Menu](#)

Digitize

The Digitize function is used with the **MultiCam** Rapid Shape™ feature. With this feature, the operator has the ability to duplicate 2D simple and complex shapes on the machine and then import this outline into a CAD program for manipulation and toolpathing. Not all **MultiCam** systems carry the Digitize option, but the Digitize option can be purchased from the local distributor.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until Digitize is highlighted in blue.
3. Press **Enter**  to access the Digitize sub-menu.
4. Press **X-axis Jog Arrows**  until the menu item is highlighted in blue. Some menu items will then enter a sub-menu with more selections.



[Start Digitize](#)
[Auto Step Size](#)
[Add Corners](#)

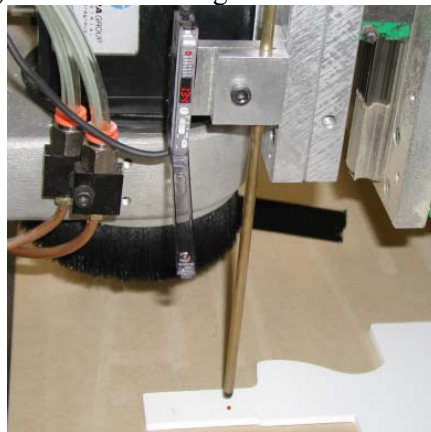
[Auto Scan Rate](#)
[Scan Mode](#)
[Output Start](#)

[Menu](#)

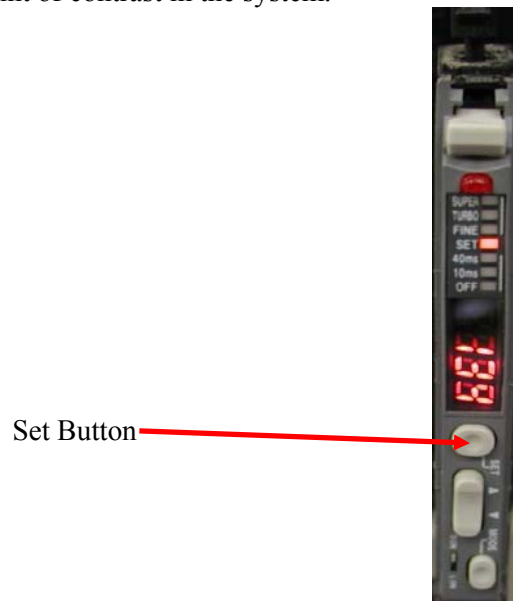
Start Digitize

The Start Digitize function configures the system to begin 2D digitizing. Operators **should not use** this menu option until all other Digitize options have been set.

1. Place the object to be scanned on a contrasting sacrifice material. A white object on a dark background will have a better contrast than a wooden object on MDF.
2. Turn on the table vacuum to hold the part flat to the sacrifice material. Warped material can introduce errors in the scanning process.
3. Lower the LED probe to usually about 3/8" – 1/2" above the material. The numbers on the counter will decline (e.g., 70 → 134 and then 134 → 16) at the contrast edge.



4. Raise the probe until the counter shows the higher number (e.g., 134) and press the Set button. This will save the highest point of contrast in the system.



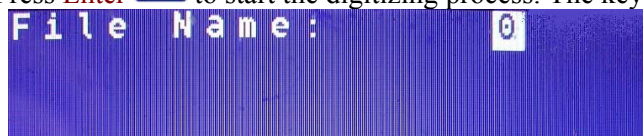
5. Press **Menu**  to access the menu system.

6. Press **X-axis Jog Arrows**   until Digitize is highlighted in blue.

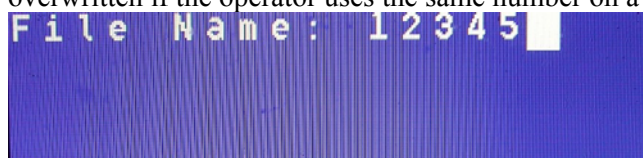
7. Press **Enter**  to access the Digitize sub-menu.

8. Press **X-axis Jog Arrows**   until Start Digitize is highlighted in blue.

9. Press **Enter**  to start the digitizing process. The keypad will prompt the operator for the file name.



10. Enter a numerical file name (e.g., 12345) and press **Enter** . This number is arbitrary and can be overwritten if the operator uses the same number on a different digitize file.



11. Jog the digitizer laser to the beginning of the object being scanned.



12. Place the dot over the edge of the target and non-target areas and press **Enter** . This will set the point for the file.

13. Jog the digitizer laser away slightly so that the beam splits the target and non-target areas.

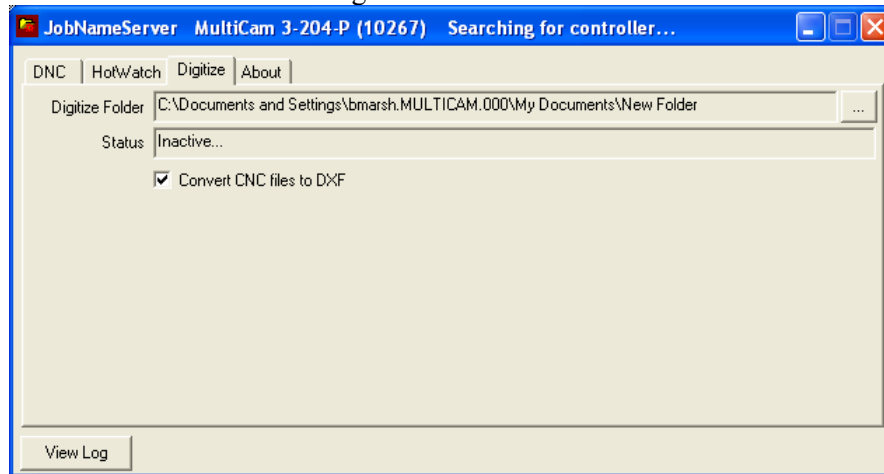
14. Press **Five**  to begin Auto Scan. The system will scan back and forth across the object for a few seconds at the rate and step size entered in the Digitize menu items before moving in the set direction along the contour of the pattern. The scanner will continue around the pattern until the start point is reached unless interrupted by the user.

- If the probe is too far away from the object, then the display will indicate that the edge of the part could not be found, and operators will need to adjust the probe.
- If the Auto Scan Rate is too high, then the Rapid Shape could shake and lose signal.

15. Press **Cancel**  to finish scanning and exit out of Digitize mode.

16. Input the completed scanned file in EnRoute and clean up as needed.

17. Open Machine Tools > JobNameServer > Digitize.



18. Click on  and select the folder to save the file. All files are saved automatically as CNC files. Operators who want to save the file as a DXF should make sure the checkbox is filled.

[Digitize](#)

[Menu](#)

Auto Scan Rate

The Auto Scan Rate function sets the automatic rate for the scanning process during the 2D digitizing. The system defaults to 5" per second, but the normal recommended scan rate is 1" per second. Slower scan rates will be more accurate but will take longer to complete whereas faster scan rates will be less accurate but will take less time to complete. Operators will need to rely on the material, the complexity of the object, and experience to find the best time.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until Digitize is highlighted in blue.
3. Press **Enter**  to access the Digitize sub-menu.
4. Press **X-axis Jog Arrows**  until Auto Scan Rate is highlighted in blue.
5. Press **Enter**  to access the automatic scan rate function.

6. Enter the value for the scan rate (e.g., 1.5).

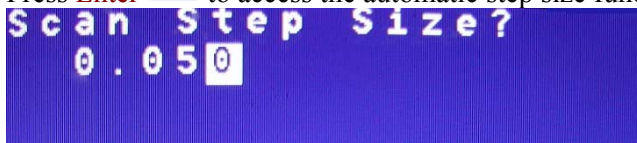
7. Press **Enter**  to save the Scan Rate and return to the Digitize sub-menu.

[Digitize](#)[Menu](#)

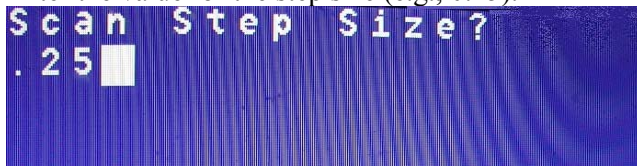
Auto Step Size

The Auto Step Size function sets the automatic distance used during the 2D digitizing. The step size can be adjusted per the part being scanned. If the 2D detail is complicated or the part is small, then operators are encouraged to use a smaller step size (e.g., 0.050" [1.27mm]). If the 2D detail is simple or the part is large, then operators are encouraged to use a larger step size (e.g., 1.0" [25.4mm]).

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**   until Digitize is highlighted in blue.
3. Press **Enter**  to access the Digitize sub-menu.
4. Press **X-axis Jog Arrows**   until Auto Step Size is highlighted in blue.
5. Press **Enter**  to access the automatic step size function.



6. Enter the value for the step size (e.g., 0.25).



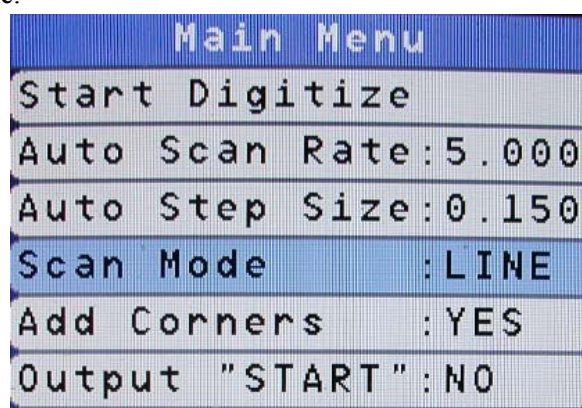
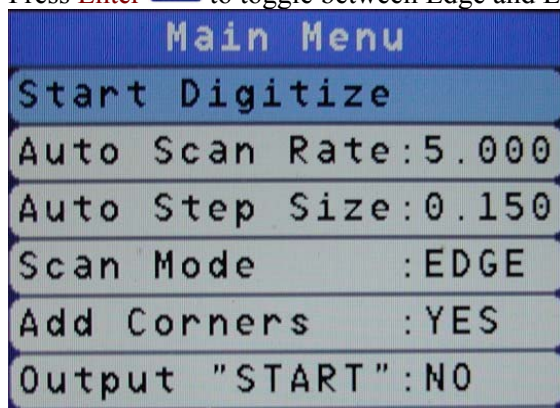
7. Press **Enter**  to save the Step Size and return to the Digitize sub-menu.

[Digitize](#)[Menu](#)

Scan Mode

The Scan Mode function determines the edge or line mode for scanning. Operators can use EDGE for solids and LINE for outlines.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until Digitize is highlighted in blue.
3. Press **Enter**  to access the Digitize sub-menu.
4. Press **X-axis Jog Arrows**  until Scan Mode is highlighted in blue.
5. Press **Enter**  to toggle between Edge and Line.

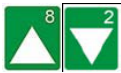


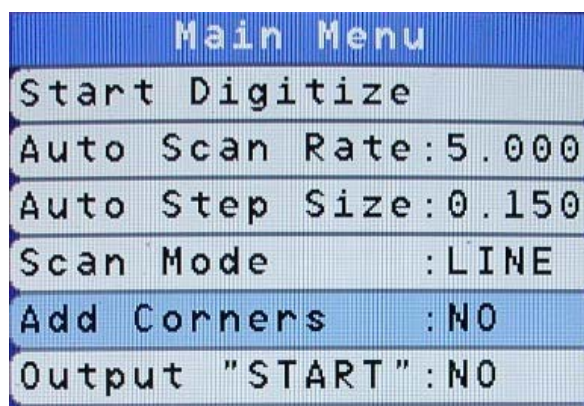
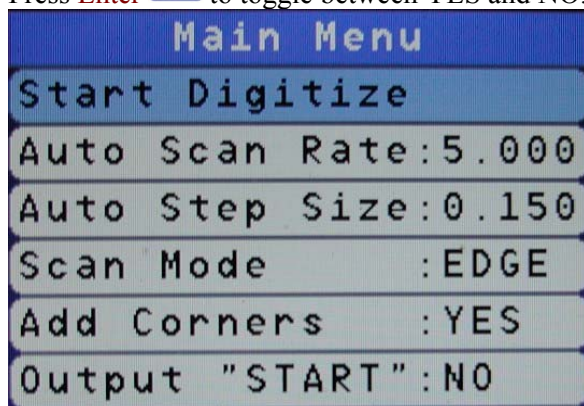
6. Continue with the Digitize setup or press **Cancel**  to exit the Digitize sub-menu.

[Digitize](#)[Menu](#)

Add Corners

The Add Corners function accentuates defined corners at any major direction change when ON and rolls corners when OFF. Operators should turn this setting ON if the part being scanned has several sharp corners and can leave the setting OFF if most of the corners are rounded.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until Digitize is highlighted in blue.
3. Press **Enter**  to access the Digitize sub-menu.
4. Press **X-axis Jog Arrows**  until Add Corners is highlighted in blue.
5. Press **Enter**  to toggle between YES and NO.



6. Continue with the Digitize setup or press **Cancel**  to exit the Digitize sub-menu.

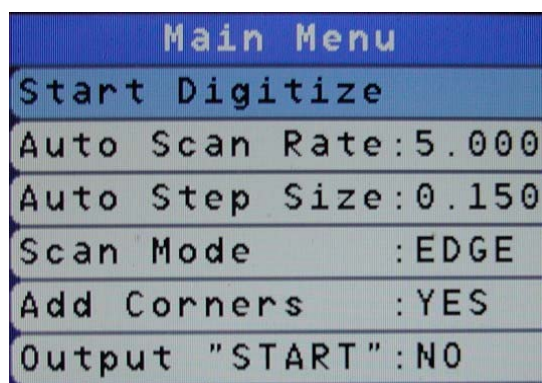
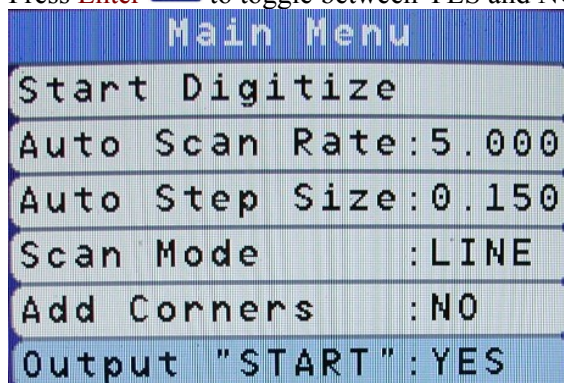
[Digitize](#)

[Menu](#)

Output Start

The Output Start function signals the machine to add the word “Start” to the beginning of the code used in the cutting sequence. This code may be required in some CAD programs for importation.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until Digitize is highlighted in blue.
3. Press **Enter**  to access the Digitize sub-menu.
4. Press **X-axis Jog Arrows**  until Output Start is highlighted in blue.
5. Press **Enter**  to toggle between YES and NO.

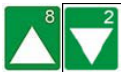


6. Continue with the Digitize setup or press **Cancel**  to exit the Digitize sub-menu.

[Digitize](#)[Menu](#)

Shutdown

The Shutdown function takes the servos offline to allow the machine to be powered down safely.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until Shutdown is highlighted in blue.
3. Press **Enter**  to access the Shutdown sequence. Operators may press **Y-axis Jog Arrows**  to move between Yes and No.



4. Press **Enter**  when Yes is highlighted to shut down the machine. The system will disable the motor drives and prompt the operator when it is safe to power down the machine. If No is selected, then the display will return to the main menu.

[Menu](#)

Cut Utilities

The Cut Utilities function allows the operator to cut common shapes out of material. All options but Table Mill ignore the diameter of the bit (e.g., for 10" square cut with 1/2" bit, operator must input 10.5" x 10.5").

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until Cut Utilities is highlighted in blue.
3. Press **Enter**  to access the Cut Utilities sub-menu.
4. Press **X-axis Jog Arrows**  until the menu item is highlighted in blue. Some menu items will then enter a sub-menu with more selections.

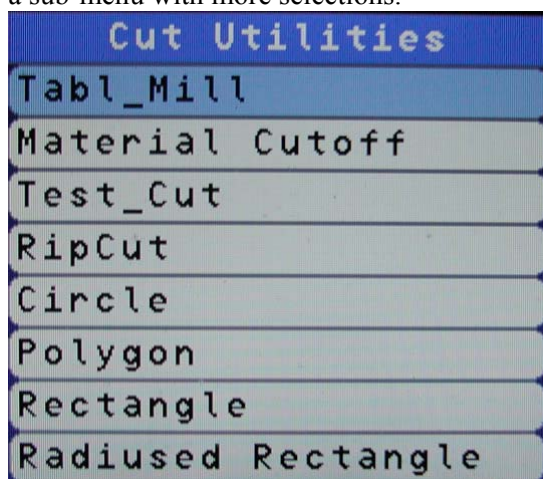
[Tabl_Mill](#)[Material Cutoff](#)[Test_Cut](#)[RipCut](#)[Circle](#)[Polygon](#)[Rectangle](#)[Radiused Rectangle](#)[Menu](#)

Table Mill

The Table Mill function is used to surface or resurface the tabletop or any other piece of material that needs to be flat. The program is designed to use any fly cutter.

1. Set up the system as if a normal file is to be executed (e.g., Set Home, Set Surface, Tool Lift, Cut Depth, X & Y Feedrate, and Z Plunge Feed).

2. Set home in the lower right hand corner of the table or the material to be milled.

3. Press **Jog Arrows**  to position the cutter in the upper left hand corner of the table or the material to be milled.

4. Press **Menu**  to access the menu system.

5. Press **X-axis Jog Arrows**  until Cut Utilities is highlighted in blue.

6. Press **Enter**  to enter the Cut Utilities sub-menu.

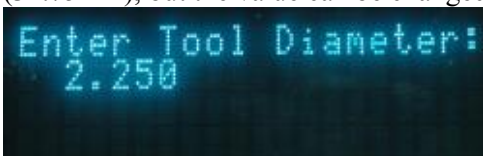
7. Press **X-axis Jog Arrows**  until Tabl_Mill is highlighted in blue.

8. Press **Enter**  access the Table Mill function.

9. Enter a tool number using the number keypad (e.g., 8) and press **Enter** .



10. Enter the tool diameter using the number keypad if a different size is required. The default value is 1.25" (31.75mm), but the value can be changed for a different size cutter.



11. Press **Enter**  to accept the value. Operators may press **X-axis Down Jog Arrow**  to change Tool Parameters if necessary.

12. Press **Start**  to execute Table Mill, **Cancel**  to abort the function, or **Pause**  to reposition the spindle start point.



The spindle will start, and the machine will begin to mill the table surface. The overlap between passes is 20% of the tool diameter.

[Cut Utilities](#) [Menu](#)

Table Mill Tool Parameters

The Table Mill Tool Parameters function is used to specify the tool diameter and add lead ins and lead outs.

1. Press **X-axis Down Jog Arrow**  from the Table Mill Start/Cancel/Pause screen.



2. Press **Enter**  to adjust the Table Mill Tool Parameters.



a. Tool Diameter

- Press **Enter**  over Tool Diam.
- Enter the corrected value using the number keypad. Negative numbers should be used for an inside cut.
- Press **Enter**  to confirm the value and return to the Table Mill Tool Parameters sub-menu.

b. Lead In/Out

- Press **Enter**  over Lead In.
- Press **Enter**  to toggle between YES and NO.

c. Lead In Radius

- Press **Enter**  over Lead In Rad.
- Enter the Lead In Radius value using the number keypad.
- Press **Enter**  to confirm the value and return to the Table Mill Tool Parameters sub-menu.

[Cut Utilities](#) [Menu](#)

Material Cutoff

The Material Cutoff function allows the operator to manually cut off the remainder of the material based on X and Y input. This function does not work from a Soft Home but starts at the current Jog location and ends when the operator stops the motion

1. Set up the system as if a normal file is to be executed (e.g., Set Surface, Tool Lift, Cut Depth, X & Y Feedrate, and Z Plunge Feed). It is not necessary to set home as the cut will begin at the current X,Y location of the spindle.

2. Press **Jog Arrows**  to position the cutter in the desired start position.

3. Press **Menu**  to access the menu system.

4. Press **X-axis Jog Arrows**  until Cut Utilities is highlighted in blue.

5. Press **Enter**  to access the Cut Utilities sub-menu.

6. Press **X-axis Jog Arrows**  until Matl_Cut is highlighted in blue.

7. Press **Enter**  to access the Material Cutoff function.

8. Enter the desired tool number if different from the one currently loaded and press **Enter** .



9. Enter the desired value for the X Cutoff Distance using the number keypad (e.g., 0) and press **Enter** .



10. Enter the desired value for the Y Cutoff Distance using the number keypad (e.g., 60) and press **Enter** .



11. Press **Start**  to execute Material Cutoff, **Cancel**  to abort the function, or **Pause**  to reposition the spindle start point.



[Cut Utilities](#)

[Menu](#)

Test Cut

The Test Cut function is used to run a sample file to test the output of the system. A 0.25" end mill or small engraving cutter can be used to execute the test cut file. If a 0.25" bit is used for the test cut, then the machine will cut a 0.5" circle, 1" diamond, and a 1.5" square.

1. Set up the system as if a normal file is to be executed (e.g., Set Home, Set Surface, Tool Lift, Cut Depth, X & Y Feedrate, and Z Plunge Feed) and make sure the spindle is enabled.

2. Press **Menu**  to access the menu system.

3. Press **X-axis Jog Arrows**   until Cut Utilities is highlighted in blue.

4. Press **Enter**  to access the Cut Utilities sub-menu.

5. Press **X-axis Jog Arrows**   until Test_Cut is highlighted in blue.

6. Press **Enter**  to access the Test Cut function. Operators may press **X-axis Down Jog Arrow**  to change Tool Parameters if necessary.

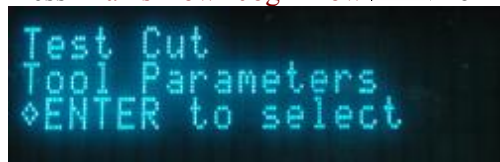
7. Press **Start**  to execute Test Cut, **Cancel**  to abort the function, or **Pause**  to reposition the spindle start point.

[Cut Utilities](#)[Menu](#)

Test Cut Tool Parameters

The Test Cut Tool Parameters function is used to specify the tool diameter and to add lead ins and lead outs.

1. Press **X-axis Down Jog Arrow**  from the Test Cut Start/Cancel/Pause screen.



2. Press **Enter**  to adjust the Test Cut Tool Parameters.



a. Tool Diameter

- i. Press **Enter**  over Tool Diam.
- ii. Enter the corrected value using the number keypad. Negative numbers should be used for an inside cut.
- iii. Press **Enter**  to confirm the value and return to the Test Cut Tool Parameters sub-menu.

b. Lead In/Out

- i. Press **Enter**  over Lead In.
- ii. Press **Enter**  to toggle between YES and NO.

c. Lead In Radius

- i. Press **Enter**  over Lead In Rad.
- ii. Enter the Lead In Radius value using the number keypad.
- iii. Press **Enter**  to confirm the value and return to the Test Cut Tool Parameters sub-menu.

[Cut Utilities](#) [Menu](#)

Rip Cut

The Rip Cut function will allow the operator to manually cut wherever the machine is Jogged so that the operator can remove excess material. This function does not work from Soft Home.

1. Set up the system as if a normal file is to be executed (e.g., Set Surface, Tool Lift, Cut Depth, X & Y Feedrate, and Z Plunge Feed). It is not necessary to set home as the cut will begin at the current X,Y location of the spindle.

2. Press **Jog Arrows**  to position the cutter in the desired start position.

3. Press **Menu**  to access the menu system.

4. Press **X-axis Jog Arrows**  until Cut Utilities is highlighted in blue.

5. Press **Enter**  to access the Cut Utilities sub-menu.

6. Press **X-axis Jog Arrows**  until RipCut is highlighted in blue.

7. Press **Enter**  to access the Rip Cut function.



8. Press **Set Surface**  to toggle between Jog and Cut.



9. Press **Jog Arrows**  to start moving the spindle, which will plunge into the material after reaching the programmed RPM. The spindle will cut for as long as the Jog button is depressed.

10. Press **Set Surface**  again and as needed to toggle between Jog and Cut

11. Press **Cancel**  to exit the Rip Cut mode.

[Cut Utilities](#)

[Menu](#)

Circle

The Circle function executes any size circle shape. Home is located at the center, and the cutting head moves out the length of the radius to complete the cut. Operators should verify that there is enough material to complete the circle before beginning the file.

1. Set up the system as if a normal file is to be executed (e.g., Set Home, Set Surface, Tool Lift, Cut Depth, X & Y Feedrate, and Z Plunge Feed) and make sure the spindle is enabled.

2. Press **Menu**  to access the menu system.

3. Press **X-axis Jog Arrows**   until Cut Utilities is highlighted in blue.

4. Press **Enter**  to access the Cut Utilities sub-menu.

5. Press **X-axis Jog Arrows**   until Circle is highlighted in blue.

6. Press **Enter**  to access the Circle function.

7. Enter the radius of the Circle on the display using the number keypad (e.g., 0.50) and press **Enter** .



Operators may press **X-axis Down Jog Arrow**  to change Tool Parameters if necessary.

8. Press **Start**  to execute Circle Cut, **Cancel**  to abort the function, or **Pause**  to reposition the spindle start point.



[Cut Utilities](#)

[Menu](#)

Circle Shape Tool Parameters

The Circle Shape Tool Parameters function is used to specify the tool diameter and to add lead ins and lead outs.

1. Press **X-axis Down Jog Arrow**  from the Circle Shape Start/Cancel/Pause screen.



2. Press **Enter**  to access the Circle Shape Tool Parameters.



a. Tool Diameter

- i. Press **Enter**  over Tool Diam.
- ii. Enter the corrected value using the number keypad. Negative numbers should be used for an inside cut.
- iii. Press **Enter**  to confirm the value and return to the Circle Shape Tool Parameters sub-menu.

b. Lead In/Out

- i. Press **Enter**  over Lead In.
- ii. Press **Enter**  to toggle between YES and NO.

c. Lead In Radius

- i. Press **Enter**  over Lead In Rad.
- ii. Enter the Lead In Radius value using the number keypad.
- iii. Press **Enter**  to confirm the value and return to the Circle Shape Tool Parameters sub-menu.

[Cut Utilities](#) [Menu](#)

Polygon

The Polygon function executes any size polygon shape based on number of sides and radius input by operator. Home is located at the center, and the cutting head moves out the length of the radius to complete the cut. Operators should verify that there is enough material to complete the polygon before beginning the file.

1. Set up the system as if a normal file is to be executed (e.g., Set Home, Set Surface, Tool Lift, Cut Depth, X & Y Feedrate, and Z Plunge Feed) and verify that the spindle is enabled.

2. Press **Menu**  to access the menu system.

3. Press **X-axis Jog Arrows**   until Cut Utilities is highlighted in blue.

4. Press **Enter**  to access the Cut Utilities sub-menu.

5. Press **X-axis Jog Arrows**   until Polygon is highlighted in blue.

6. Press **Enter**  to access the Polygon function.

7. Enter the number of sides for the Polygon using the number keypad (e.g., 3) and press **Enter** .



8. Enter the radius of the Polygon using the number keypad (e.g., 0.5) and press **Enter** .



Operators may press **X-axis Down Jog Arrow**  to change Tool Parameters if necessary.

9. Press **Start**  to execute Polygon Cut, **Cancel**  to abort the function, or **Pause**  to reposition the spindle start point.



[Cut Utilities](#)

[Menu](#)

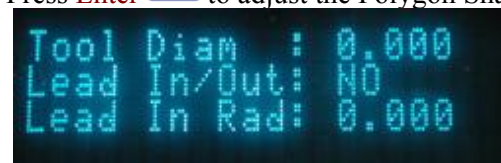
Polygon Shape Tool Parameters

The Polygon Shape Tool Parameters function is used to specify the tool diameter and to add lead ins and lead outs.

1. Press **X-axis Down Jog Arrow**  from the Polygon Shape Start/Cancel/Pause screen.



2. Press **Enter**  to adjust the Polygon Shape Tool Parameters.



a. Tool Diameter

- Press **Enter**  over Tool Diam.
- Enter the corrected value using the number keypad. Negative numbers should be used for an inside cut.
- Press **Enter**  to confirm the value and return to the Polygon Shape Tool Parameters sub-menu.

b. Lead In/Out

- Press **Enter**  over Lead In.
- Press **Enter**  to toggle between YES and NO.

c. Lead In Radius

- Press **Enter**  over Lead In Rad.
- Enter the Lead In Radius value using the number keypad.
- Press **Enter**  to confirm the value and return to the Polygon Shape Tool Parameters sub-menu.

[Cut Utilities](#)

[Menu](#)

Rectangle

The Rectangle function executes any length and width size rectangle shape based on operator input. Home is located at the front right corner of the shape.

1. Set up the system as if a normal file is to be executed (e.g., Set Home, Set Surface, Tool Lift, Cut Depth, X & Y Feedrate, and Z Plunge Feed) and make sure the spindle is enabled.

2. Press **Menu**  to access the menu system.

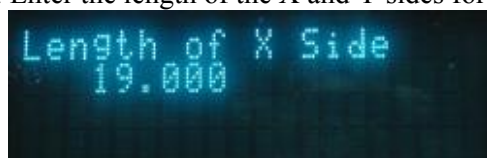
3. Press **X-axis Jog Arrows**   until Cut Utilities is highlighted in blue.

4. Press **Enter**  to access the Cut Utilities sub-menu.

5. Press **X-axis Jog Arrows**   until Rectangle is highlighted in blue.

6. Press **Enter**  to access the Rectangle function.

7. Enter the length of the X and Y sides for the Rectangle using the number keypad (e.g., X=19, Y=20).



8. Press **Enter**  to accept the value.



9. Press **Enter**  to accept the value. Operators may press **X-axis Down Jog Arrow**  to change Tool Parameters if necessary.

10. Press **Start**  to execute Rectangle Cut, **Cancel**  to abort the function, or **Pause**  to reposition the spindle start point.



[Cut Utilities](#)

[Menu](#)

Rectangle Shape Tool Parameters

The Rectangle Shape Tool Parameters function is used to specify the tool diameter and to add lead ins and lead outs.

1. Press **X-axis Down Jog Arrow**  from the Rectangle Shape Start/Cancel/Pause screen.



2. Press **Enter**  to adjust the Rectangle Shape Tool Parameters.



a. Tool Diameter

- i. Press **Enter**  over Tool Diam.
- ii. Enter the corrected value using the number keypad. Negative numbers should be used for an inside cut.
- iii. Press **Enter**  to confirm the value and return to the Rectangle Shape Tool Parameters sub-menu.

b. Lead In/Out

- i. Press **Enter**  over Lead In.
- ii. Press **Enter**  to toggle between YES and NO.

c. Lead In Radius

- i. Press **Enter**  over Lead In Rad.
- ii. Enter the Lead In Radius value using the number keypad.
- iii. Press **Enter**  to confirm the value and return to the Rectangle Shape Tool Parameters sub-menu.

[Cut Utilities](#) [Menu](#)

Radiused Rectangle

The Radiused Rectangle function executes any radius, length, and width size rectangle shape based on operator input. Home is located at the front right corner of the shape.

1. Set up the system as if a normal file is to be executed (e.g., Set Home, Set Surface, Tool Lift, Cut Depth, X & Y Feedrate, and Z Plunge Feed) and make sure the spindle is enabled.

2. Press **Menu**  to access the menu system.

3. Press **X-axis Jog Arrows**  until Cut Utilities is highlighted in blue.

4. Press **Enter**  to access the Cut Utilities sub-menu.

5. Press **X-axis Jog Arrows**  until Radiused Rectangle is highlighted in blue.

6. Press **Enter**  to access the Radiused Rectangle function.

7. Enter the length of the X and Y sides for the Rectangle using the number keypad (e.g., X=19, Y=20).



8. Press **Enter**  to accept the value.



9. Press **Enter**  to accept the value.

10. Enter the radius of the corners using the number keypad (e.g., 1.5).



11. Press **Enter**  to accept the value of the radius. Operators may press **X-axis Down Jog Arrow**  to change Tool Parameters if necessary.

12. Press **Start**  to execute Radiused Rectangle Cut, **Cancel**  to abort the function, or **Pause**  to reposition the start point.



```
Radiused Rectangle
♦START to Execute
♦CANCEL to Abort
♦PAUSE to Yield
```

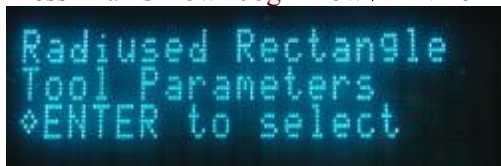
[Cut Utilities](#)

[Menu](#)

Radiused Rectangle Shape Tool Parameters

The Radiused Rectangle Shape Tool Parameters function is used to specify the tool diameter and add lead ins and lead outs.

1. Press **X-axis Down Jog Arrow**  from the Radiused Rectangle Shape Start/Cancel/Pause screen.



2. Press **Enter**  to adjust the Radiused Rectangle Shape Tool Parameters.



a. Tool Diameter

- i. Press **Enter**  over Tool Diam.
- ii. Enter the corrected value using the number keypad. Negative numbers should be used for an inside cut.
- iii. Press **Enter**  to confirm the value and return to the Radiused Rectangle Shape Tool Parameters sub-menu.

b. Lead In/Out

- i. Press **Enter**  over Lead In.
- ii. Press **Enter**  to toggle between YES and NO.

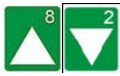
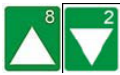
c. Lead In Radius

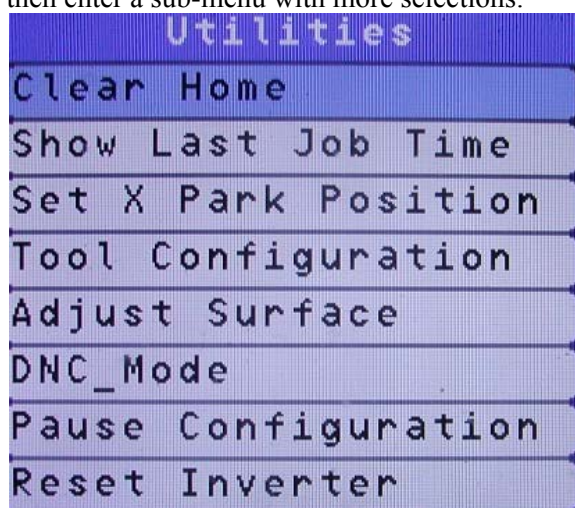
- i. Press **Enter**  over Lead In Rad.
- ii. Enter the Lead In Radius value using the number keypad.
- iii. Press **Enter**  to confirm the value and return to the Radiused Rectangle Shape Tool Parameters sub-menu.

[Cut Utilities](#) [Menu](#)

Utility

The Utility sub-menu is used to provide the operator with additional options for completing machine operations. Some of these menu items contain a sub-menu for further specifications.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until Utility is highlighted in blue.
3. Press **Enter**  to access the Utility sub-menu.
4. Press **X-axis Jog Arrows**  until the appropriate option is highlighted in blue. Certain menu items will then enter a sub-menu with more selections.



Clear Home	Show Last Job Time
Set X Park Position	Tool Configuration
Adjust Surface	DNC_Mode
Pause Configuration	Reset Inverter
Reboot	Lube
Dust Collector	Default Max Depth
MisterOpt	No Spin
SPA Trigg	GDrillTest
Dust Collector Test	

[Menu](#)

Clear Home

The Clear Home function is used to displays the X & Y coordinates of the Soft Home position from the Hard Home location. Clearing Home does not erase the Soft Home.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**   until Utility is highlighted in blue.
3. Press **Enter**  to access the Utility sub-menu.
4. Press **X-axis Jog Arrows**   until Clear Home is highlighted in blue.
5. Press **Enter**  to clear the Home position and return to the Utility sub-menu.

[Utility](#)[Menu](#)

Show Last Job Time

The Show Last Job Time function is used to display the amount of time the previous job required.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**   until Utility is highlighted in blue.
3. Press **Enter**  to access the Utility sub-menu.
4. Press **X-axis Jog Arrows**   until Show Last Job Time is highlighted in blue.
5. Press **Enter**  to view the exact job time.

[Utility](#)[Menu](#)

Set X Park Position

The Set X Park Position function is used to set a specific Park location for the X-axis to assume after each job, which is useful when currently using only a portion of the available table space. The gantry can be set to move just far enough out of the way for the material to be removed instead of parking all the way to the rear every time.

1. Press **Menu**  to access the menu system.

2. Press **X-axis Jog Arrows**   until Utility is highlighted in blue.

3. Press **Enter**  to access the Utility sub-menu.

4. Press **X-axis Jog Arrows**   until Set X Park Position is highlighted in blue.

5. Press **Enter**  to access the Set X Park Position function.

6. Enter the desired park position for the spindle in X or enter 0 to use the table size.



```
Enter X Max Park
Position?
0.0 = use table size
100.000
```

7. Press **Enter**  to accept the position and return to the Utility sub-menu.

[Utility](#)

[Menu](#)

Tool Configuration

The Tool Configuration function allows the operator to set the individual characteristics of each tool up to the maximum number of tools allowed for the machine type (e.g., Rotary ATC = 50, Linear ATC = 25). The operator can set each tool to recognize a dust collector, mister, spin function, and pause Z lift during the cut and traverse sequences. This menu item is only available on routers configured with an ATC.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until Utility is highlighted in blue.
3. Press **Enter**  to access the Utility sub-menu.
4. Press **X-axis Jog Arrows**  until Tool Configuration is highlighted in blue.
5. Press **Jog Arrows**  to reach the tool and particular component to be configured.

Tool Configuration				
Tool	Dust	Mist	Spin	PLift
1	Y	No	Y	Y
2	Y	No	Y	Y
3	Y	No	Y	Y
4	Y	No	Y	Y
5	Y	No	Y	Y
6	Y	No	Y	Y
7	Y	No	Y	Y

6. Press **Enter**  to toggle each cell from Y to No. The system will default to all tools showing Y for Dust, Spin, and PLift and No for Mist. Operators may select No for Spin when working with tools that do not need rotation for the cut (e.g., drag knife). The PLift should be set to No for operators who believe that the lifting of the Z during pause would destroy the material or quality of the cut (e.g., T-slot cutter).

- Dust – Dust Collector active during the cutting sequence
- Mist – Mister active during the cutting sequence
- Spin – Spin of tool active during the cutting sequence
- PLift – Z Pause Lift active during the cutting sequence

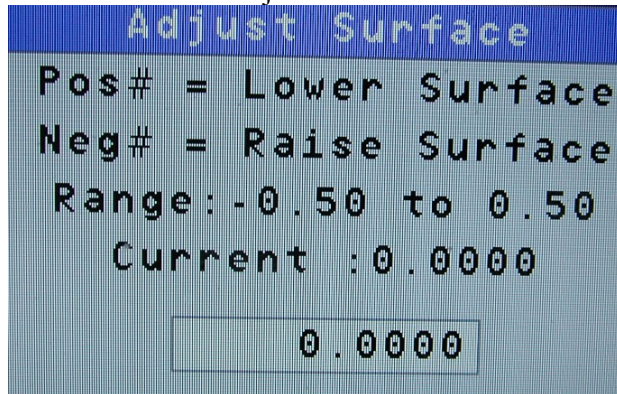
7. Press **Cancel**  to return to the Utility sub-menu.

[Utility](#)
[Menu](#)

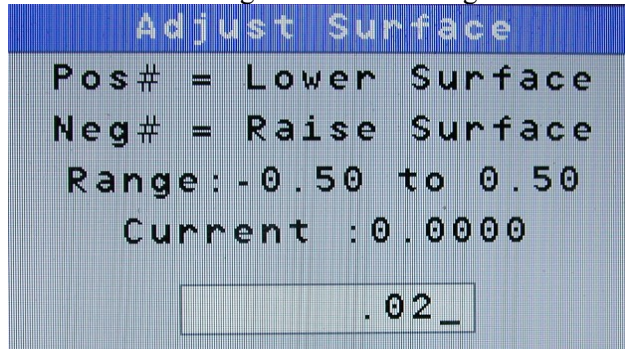
Adjust Surface

The Adjust Surface function is used to adjust the surface setting without having to go through the surfacing routine. The surface **must already be set** before it can be adjusted.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until Utility is highlighted in blue.
3. Press **Enter**  to access the Utility sub-menu.
4. Press **X-axis Jog Arrows**  until Adjust Surface is highlighted in blue.
5. Press **Enter**  to adjust the surface.



6. Enter the value for the value for the adjustment (e.g., 0.02). A positive number moves the surface closer to the table bed while a negative number brings the surface closer to the cutting head.



7. Press **Enter**  to accept the setting and return to the Utility sub-menu.

[Utility](#)

[Menu](#)

DNC Mode

The DNC Mode function is used to allow the operator to toggle back and forth between local and remote DNC Modes. Local mode allows the operator to review what files have been saved in the controller, and remote mode allows the operator to run programs. The system must be set to remote mode to run the file.

The DNC Mode can create a shortcut for commonly used files without the operator having to enter the Utility sub-menu each time. The controller memory allows a maximum of 20 job files or a limited amount of memory space for large job files to be saved to the controller. The display will indicate when the memory capacity is full with a notice showing *Error Writing File; Out of Room*. The operator will need to delete a job file from the controller memory before adding any new job files.

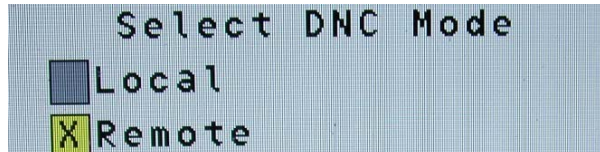
1. Press **Menu**  to access the menu system.

2. Press **X-axis Jog Arrows**  until Utility is highlighted in blue.

3. Press **Enter**  to access the Utility sub-menu.

4. Press **X-axis Jog Arrows**  until DNC Mode is highlighted in blue.

5. Press **Enter**  to access the DNC mode. An X will be located to the left of the mode currently being run.

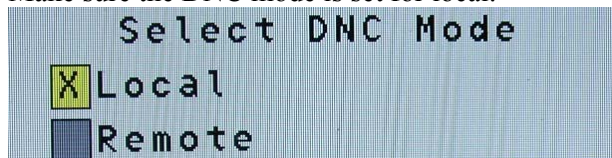


6. Press **X-axis Jog Arrows**  to move between the DNC Modes. Local mode shows what files have been saved to the controller while the remote mode shows what files are available in the PC.

7. Press **Enter**  to accept the selected mode and return to the Utility sub-menu.

Saving Files to Controller DNC

1. Make sure the DNC mode is set for local.

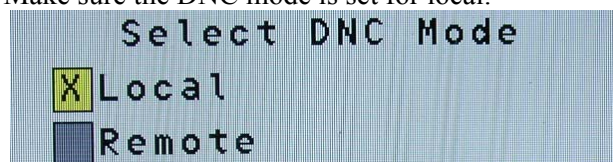


2. Press **Shift**  and **DNC**  to access the list of files available in the PC.

3. Press **X-axis Jog Arrows**   until the particular job file to be saved to the controller is highlighted in blue.
4. Press **Enter**  to accept the job file.
5. Press **Start**  to save the file to the controller. The operator can access the file from the **DNC**  function at the keypad.
6. Press **Cancel**  to return to the Utility sub-menu.
7. Press **DNC**  to access the files available from the controller.
8. Press **X-axis Jog Arrows**   until the particular job file to be run is highlighted in blue.
9. Press **Enter**  to access the job file and then **Start**  to execute the job file.

Deleting Files from Controller DNC

1. Make sure the DNC mode is set for local.

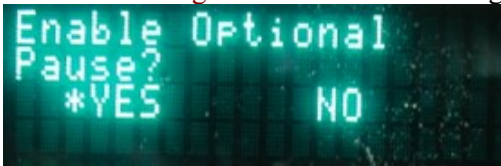


2. Press **DNC**  to access the list of job files available in the controller.
3. Press **X-axis Jog Arrows**   until the particular job file to be deleted from the controller is highlighted in blue.
4. Press **Enter**  to access the job file.
5. Press **Set Max Depth**  to set the system to delete the job file.
6. Press **Enter**  to delete the job file.

[Utility](#)[Menu](#)

Pause Configuration

The Pause Configuration function allows operators to disregard the pause called out in the M Code (i.e., M01) usually included for testing purposes.

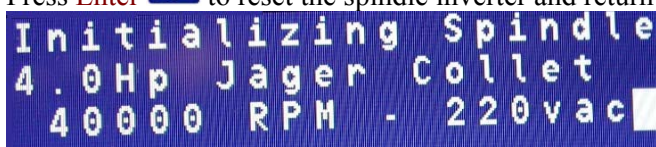
1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until Utility is highlighted in blue.
3. Press **Enter**  to access the Utility sub-menu.
4. Press **X-axis Jog Arrows**  until Pause Configuration is highlighted in blue.
5. Press **Enter**  to access the Pause Configuration setting.
6. Press **Y-axis Jog Arrows**  to toggle between Yes and No.

7. Press **Cancel**  to exit Pause Configuration.

[Utility](#)[Menu](#)

Reset Inverter

The Reset Inverter function is used to reset the spindle inverter through MODBUS.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**   until Utility is highlighted in blue.
3. Press **Enter**  to access the Utility sub-menu.
4. Press **X-axis Jog Arrows**   until Reset Inverter is highlighted in blue.
5. Press **Enter**  to reset the spindle inverter and return to the Utility menu.

[Utility](#)[Menu](#)

Reboot

The Reboot function is used to restart the machine without having to power down the machine.

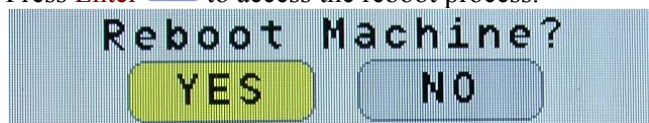
1. Press **Menu**  to access the menu system.

2. Press **X-axis Jog Arrows**  until Utility is highlighted in blue.

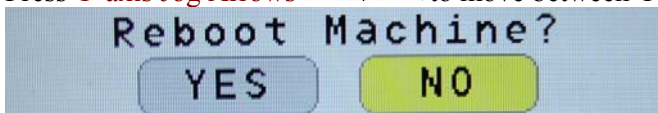
3. Press **Enter**  to access the Utility sub-menu.

4. Press **X-axis Jog Arrows**  until Reboot is highlighted in blue.

5. Press **Enter**  to access the reboot process.



6. Press **Y-axis Jog Arrows**  to move between Yes and No.



7. Press **Enter**  to process the reboot of the machine. If No is selected, then the display will return to the Utility sub-menu.

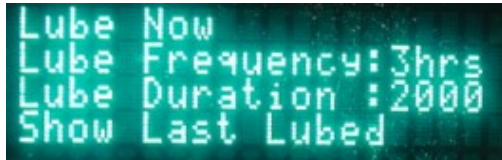
[Utility](#)

[Menu](#)

Lube

The Lube function has 4 separate sub-functions that relate to the lubrication process on machines that have the auto mister installed.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until Utility is highlighted in blue.
3. Press **Enter**  to access the Utility sub-menu.
4. Press **X-axis Jog Arrows**  until Lube is highlighted in blue.
5. Press **Enter**  to access the Lube sub-menu.



6. Press **X-axis Jog Arrows**  until the appropriate item is highlighted in blue.
 - Lube Now will apply lubrication to the system.
 - Lube Frequency will allow the operator to set how often the lubrication should be applied to the system.
 - Lube Duration will specify how long the lubrication should be applied (in msec).
 - Show Last Lube will display the last time lubrication was applied to the system.

[Utility](#)[Menu](#)

Dust Collector

The Dust Collector function enters a sub-menu so the operator can manually raise and/or lower the dust collector for adjusting. If the dust collector is disabled on a specific tool, then the dust collector will automatically raise up during the tool loading and remain up during the cutting sequence. This menu option is only available on routers configured with an ATC.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**   until Utility is highlighted in blue.
3. Press **Enter**  to access the Utility sub-menu.
4. Press **X-axis Jog Arrows**   until Dust Collector is highlighted in blue.
5. Press **Enter**  to access the dust collector settings.



6. Enter the appropriate tool number and press **Enter**  to disable the dust collector. The dust collector will remain in an up position while the tool cuts the material. Operators may undo the disabled status once the cutting sequence has completed.

[Utility](#)[Menu](#)

Default Max Depth

The Default Max Depth function allows operators to set a max depth value that can be saved to the controller. This value will not need to be reset unless the default value is cleared. This menu option is only available on routers configured with an ATC.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until Utility is highlighted in blue.
3. Press **Enter**  to access the Utility sub-menu.
4. Press **X-axis Jog Arrows**  until Default Max Depth is highlighted in blue.
5. Press **Enter**  to set the default max depth value.
6. Enter the tool number (e.g., 1) for setting the default max depth and press **Enter** .

```
Set Max Depth
Use Tool #? 1
```

7. Press **0**  to autoset the max depth or lower the Z to the max depth and press **Enter** .

```
X= 0.000 Y= 0.000
Z= 0.000 Jog= MED
Setting Max Depth
0=AutoSet, ENTER=Set
```

8. Press **Enter**  to clear the default max depth or **Cancel**  to exit the menu option. Both actions will return the screen to the Utility sub-menu.

```
Clear Default Max
Depth?

ENTER=Yes, EXIT=No
```

[Utility](#)

[Menu](#)

MisterOpt

The Mister Option function allows operators to disable the auto mister during the job file. Operators will need to enable the Mister Option function for the machine to run the auto mister during any following job files. This menu item is only available on routers configured with the collet spindle.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until Utility is highlighted in blue.
3. Press **Enter**  to access the Utility sub-menu.
4. Press **X-axis Jog Arrows**  until MisterOpt is highlighted in blue.
5. Press **Enter**  to disable the auto mister.



6. Press **Y-axis Jog Arrows**  to toggle between Yes and No.



7. Press **Enter**  to save the selection and exit Mister Option.

[Utility](#)[Menu](#)

No_Spin

The No_Spin function is used to disable the spindle motor when the operator is using a vinyl knife, pen, or comparable tool. This menu item is only available when the router is configured with a collet spindle.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**   until Utility is highlighted in blue.
3. Press **Enter**  to access the Utility sub-menu.
4. Press **X-axis Jog Arrows**   until No_Spin is highlighted in blue.
5. Press **Enter**  to access the No_Spin function.

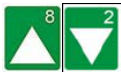


6. Enter the tool number to be set to no spin. Selecting 0 will disable the function.
7. Press **Enter**  to accept the tool number and return to the Utility menu.

[Utility](#)[Menu](#)

SPA Trigg

The Spindle Amperage Trigger (SPA Trigg) function allows the operator to check the quality of the cutting tool after a set time of consistent cutting to determine, based on the operator's observations of sharp vs. dull, if any changes should be made (e.g., sharpening, alternating) to prevent any unnecessary wear on the machine. The operator has the option to disable this function by entering 0 at the appropriate prompt.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until Utility is highlighted in blue.
3. Press **Enter**  to access the Utility sub-menu.
4. Press **X-axis Jog Arrows**  until SPA Trigg is highlighted in blue.
5. Press **Enter**  to access the SPA Trigg sub-menu.



6. Position the cursor over the appropriate parameter to set up the SPA Trigg and press **Enter** .
 - a. Enter the maximum current level in amps using the number keypad when setting the SPA Level or enter the time at the max current in seconds when setting the SPA Time.
 - b. Disable this function by selecting *SPA Time* and entering 0.

[Utility](#)[Menu](#)

GDrillTest

The Gang Drill Test function is used to perform a test of the Gang Drill. The operator can open the menu item and fire individual gang drills, as well as select up/down and motor on/off to verify operations. This menu item is only available if the router is configured with a gang drill.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until Utility is highlighted in blue.
3. Press **Enter**  to access the Utility sub-menu.
4. Press **X-axis Jog Arrows**  until GDrillTest is highlighted in blue.
5. Press **Enter**  to test the Gang Drill.



6. Press **Soft Home**  to toggle the Gang Drill on and off.
7. Press **Z-axis Jog Arrows**  to raise or lower the drill bank.
8. Press the number keys to fire individual drills by number.

[Utility](#)[Menu](#)

Dust Collector Test

The Dust Collector Test function is used to perform a test of the Dust Collector. The menu item allows the operator to test the communications to the dust collector and is only available if the router is configured with a dust collector.

1. Press **Menu**  to access the menu system.

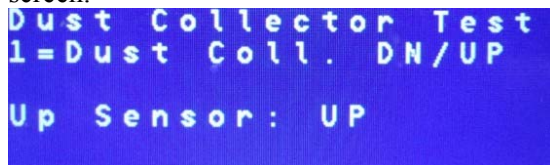
2. Press **X-axis Jog Arrows**  until Utility is highlighted in blue.

3. Press **Enter**  to access the Utility sub-menu.

4. Press **X-axis Jog Arrows**  until Dust Collector is highlighted in blue.

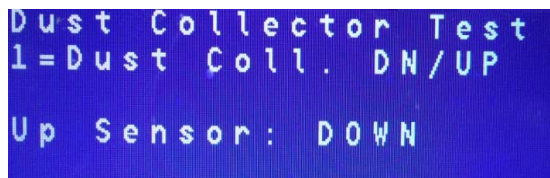
5. Press **Enter**  to test the Dust Collector.

6. Press **One**  to toggle the dust collector up and down. The active position will be listed at the bottom of the screen.



```
Dust Collector Test
1=Dust Coll. DN/UP

Up Sensor: UP
```



```
Dust Collector Test
1=Dust Coll. DN/UP

Up Sensor: DOWN
```

7. Press **Cancel**  to return to the main menu.

[Utility](#)

[Menu](#)

Tool_Comp

The Tool Compensation function is used to select a particular compensation value for a specific tool. The toolpath requires G40 commands with lead-ins/lead-outs, and the maximum compensation value is $\pm 0.100''$ (2.54mm).

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**   until Tool_Comp is highlighted in blue.
3. Press **Enter**  to access the Tool_Comp function.
4. Scroll down to the tool that will be compensated (e.g., Tool 1).

Tool_Comp		
Tool	Compensation	↑
1	0.0000	
31	0.0000	
32	0.0000	
33	0.0000	
41	0.8500	
42	0.8500	
43	0.0000	
		↓

5. Press **Enter**  and type in the necessary value (e.g., 0.025).

Tool_Comp		
Tool	Compensation	↑
1	.025_	
31	0.0000	
32	0.0000	
33	0.0000	
41	0.8500	
42	0.8500	
43	0.0000	
		↓

6. Press **Enter**  to save the value.

Tool_Comp		
Tool	Compensation	
1	0.0250	↑
31	0.0000	
32	0.0000	
33	0.0000	
41	0.8500	
42	0.8500	↓
43	0.0000	

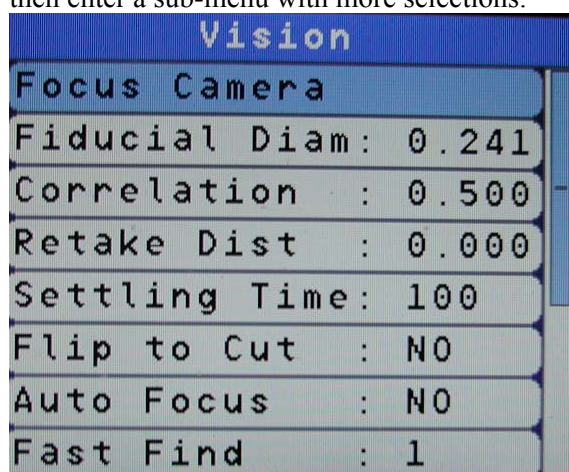
7. Press **Cancel**  to return to the main menu.

[Menu](#)

Vision

MultiVision allows operators the option of completing various projects involving repetitive cutting sequences of similar material or design. This option requires specific hardware and modified cut files. MultiVision offers a few settings that must be set once and then are stored in the controller memory until those settings are overwritten.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**   until Vision is highlighted in blue.
3. Press **Enter**  to access the Vision sub-menu.
4. Press **X-axis Jog Arrows**   until the appropriate option is highlighted in blue. Certain menu items will then enter a sub-menu with more selections.

[Focus Camera](#)[Correlation](#)[Settling Time](#)[Auto Focus](#)[Camera Delay](#)[Manual Find Setting](#)[Set Camera Offsets](#)[Fiducial Diameter](#)[Retake Distance](#)[Flip to Cut](#)[Fast Find](#)[Auto Find Setting](#)[Set Camera Scale](#)[Move to Cam Offsets](#)[Menu](#)

Focus Camera

The Focus Camera function is used to set the system to look for the fiducial, calibrate the tool if needed, and focus the camera. This enables the file to run accurately. The camera must be focused each time the material thickness changes on a standard machine. Operators should always leave the table vacuum on and use a computer that can be seen from the machine during the focusing process.

1. Calibrate the 0.25" (6.35mm) End Mill and surface the tool to the top of the material.

2. Press **Menu**  to access the menu system.

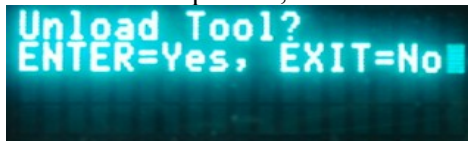
3. Press **X-axis Jog Arrows**  until Vision is highlighted in blue.

4. Press **Enter**  to access the Vision sub-menu.

5. Press **X-axis Jog Arrows**  until Focus Camera is highlighted in blue.

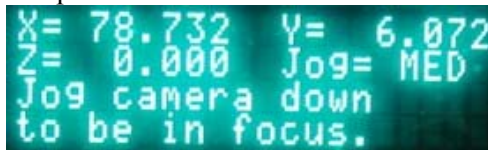
6. Press **Enter**  to unload the tool or **Cancel**  to leave the tool in the spindle if prompted. The tool may need to be unloaded depending on the length of the tool. Longer tools may interfere with the focusing process.

If **Enter**  is pressed, the machine will unload the tool and return to the material.



7. Press **Jog Arrows**  to reach the fiducial on the material to be cut.

8. Jog the camera using the **Z-axis Jog Arrows**  at the prompt to bring the fiducial into focus on the computer screen.



9. Press **Enter**  when the fiducial is in focus.

[Vision](#)

[Menu](#)

Fiducial Diameter

The Fiducial Diameter function is used to inform the system as to the size of the fiducial when the system is searching. This diameter can be set up in either inches or millimeters.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**   until Vision is highlighted in blue.
3. Press **Enter**  to access the Vision sub-menu.
4. Press **X-axis Jog Arrows**   until Fiducial Diameter is highlighted in blue.
5. Press **Enter**  to adjust the Fiducial Diameter.
6. Enter the value using the number keypad (e.g., 0.250).



7. Press **Enter**  to save the Fiducial Diameter and return to the Vision sub-menu.

[Vision](#)[Menu](#)

Correlation

The Correlation function sets a scale from 0.001 to 1.000 on the quality of the fiducial and looks at the following components:

- Shape of the fiducial on the print
- Size of the fiducial on the print compared to the size of the fiducial in the file
- X & Y location of the fiducial in the file compared to the actual location of the fiducial on the print
- Contrast between the fiducial and the background

Higher values allow the camera to accept poorer quality fiducials, some of which may not be fiducials at all but shapes on the print. The default setting of 0.2 is usually sufficient for most programs.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**   until Vision is highlighted in blue.
3. Press **Enter**  to access the Vision sub-menu.
4. Press **X-axis Jog Arrows**   until Correlation Value is highlighted in blue.
5. Press **Enter**  to access the Correlation Value function.
6. Enter the correlation value using the number keypad (e.g., 0.500).



Correlation :
0.500
Range 0.001 to 1.000

7. Press **Enter**  to save the value and return to the Vision sub-menu.

If the camera is unable to identify the fiducial, the display will prompt the operator to jog to the mark.

```
X = 45.972   Y = -10.120
Z = -5.348   Jog = FAST
Poor Correlation,
jog to mark
Press 5 for Camera View
```



Operators can press 5 to use the camera view to fully identify the fiducial. This will allow the operator to see directly on the keypad what the camera sees.

```
X = 45.972   Y = -10.120
Z = -5.348   Jog = FAST
Poor Correlation,
jog to mark
Camera View
[Image of camera view]
O Zoom      [.]
O Digitized [?]
```



Operators should press **Jog Arrows** until the camera is over the fiducial.

```
Camera View
[Image of camera view with fiducial]
O Zoom      [.]
O Digitized [?]
```

Operators can press the following keys to access specific functions:



Soft Home (.) to zoom in on the fiducial.



Help (?) to see the fiducial digitized.

```
Camera View
[Image of camera view with fiducial]
O Zoom      [.]
⊙ Digitized [?]
```

[Vision](#)
[Menu](#)

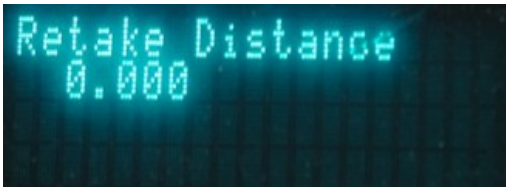
Retake Distance

The Retake Distance function allows the operator to improve the accuracy of the final cut. The camera will move out a set distance if the fiducial is not found at the programmed location.

- Low Retake Distance (e.g., 0.005) can be used if the print has multiple fiducials per image or requires a higher tolerance in the final cut. At a lower setting, the camera will take extra pictures of each fiducial.
- High Retake Distance (e.g., 0.200) can be used if the print has a limited number of fiducials per image or allows for more flexibility in the final cut. At a higher setting, the camera will cut in the bleed.

The maximum value should be 1/2 the field of view (FOV). This option becomes disabled if Fast Find is enabled.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until Vision is highlighted in blue.
3. Press **Enter**  to access the Vision sub-menu.
4. Press **X-axis Jog Arrows**  until Retake Distance is highlighted in blue.
5. Press **Enter**  to access the Retake Distance function.



6. Enter the distance for the camera to adjust so that the fiducial can be pictured clearly (e.g., 0.2" or 0.127mm).
7. Press **Enter**  to save the distance and return to the Vision sub-menu.

[Vision](#)[Menu](#)

Settling Time

The Settling Time function is used to specify how long the system must wait before reading the fiducial. This is helpful in case the camera is shaking after slewing into position. Most systems work best with 20ms. The Settling Time function applies before the picture is taken while the Camera Delay function applies after the picture is taken.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**   until Vision is highlighted in blue.
3. Press **Enter**  to access the Vision sub-menu.
4. Press **X-axis Jog Arrows**   until Settling Delay is highlighted in blue.
5. Press **Enter**  to adjust the Settling Delay.
6. Enter the value using the number keypad (e.g., 100msec).



7. Press **Enter**  to save the delay time and return to the Vision sub-menu.

[Vision](#)[Menu](#)

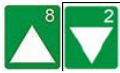
Flip to Cut

The Flip to Cut function is used to cut out a file on the back of scanned material. Operators can run the MultiVision file, flip the material, and complete the cutting sequence on the back of the material with the proper proportional cuts. In order to properly align the material for the reverse cut, operators will need to have dual sets of pop-up pins or jigs installed on their machines. Parameter 122 should be set to 0 before the Flip to Cut function can be completed.

Since the material will be flipped over, operators will need to establish the front right corner of the material as Home 8 on the right and the back of that same corner as Home 9 on the left. (The illustration below shows the Soft Home locations as they would be configured on a jig. Operators with dual pop-up pins would locate Home 8 and Home 9 in the same way shown here but within the table area defined by the pop-up pins.)



Operators should not use a Home 8 or a Home 9 to execute the file. A Soft Home should be set over the first fiducial. This will allow Flip to Cut to orient the scanned-in locations of the fiducials to the final cutout locations when the file is executed.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until Vision is highlighted in blue.
3. Press **Enter**  to access the Vision sub-menu.
4. Press **X-axis Jog Arrows**  until Flip to Cut is highlighted in blue.

5. Press **Enter**  to toggle between Yes and No.



```
Settling Time: 100
Flip to Cut   : YES
Auto Focus    : NO
Fast Find     : 1
```



```
Settling Time: 100
Flip to Cut   : NO
Auto Focus    : NO
Fast Find     : 1
```

6. Press **Cancel**  to return to the main menu. The keypad will prompt the operator to flip the material after the fiducials have been scanned in.

[Vision](#)[Menu](#)

Auto Focus

The Auto Focus function automatically moves the Z to achieve better focus at the beginning of the file execution. If the size of the fiducial is off by more than 10%, then the camera will raise or lower until the size is within 10%.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**   until Vision is highlighted in blue.
3. Press **Enter**  to access the Vision sub-menu.
4. Press **X-axis Jog Arrows**   until Auto Focus is highlighted in blue.
5. Press **Enter**  to toggle between Yes and No.



```
Settling Time: 100
Flip to Cut   : NO
Auto Focus    : YES
Fast Find     : 1
```



```
Settling Time: 100
Flip to Cut   : NO
Auto Focus    : NO
Fast Find     : 1
```

[Vision](#)[Menu](#)

Fast Find

The Fast Find function enables the camera to search for fiducials without waiting for feedback from the camera. The camera will take a picture of the fiducial and then immediately move to the next fiducial rather than interpret the data from the snapshot before moving to the next fiducial.

- Operators may enable the Fast Find function by **setting the value to 1**, which is recommended for prints with easily identifiable and high contrast fiducials, high quality presentation, and little if any skew. When Fast Find is enabled, Retake Distance is disabled.
- Operators may disable the Fast Find function by **setting the value to 0**, which is recommended for first runs of a job file or for material that is stretched or skewed.
- If the fiducial is not found while the system is in Fast Find mode, then the camera will move away from the fiducial and then **move back to the location of the fiducial** to take another picture.

1. Press **Menu**  to access the menu system.

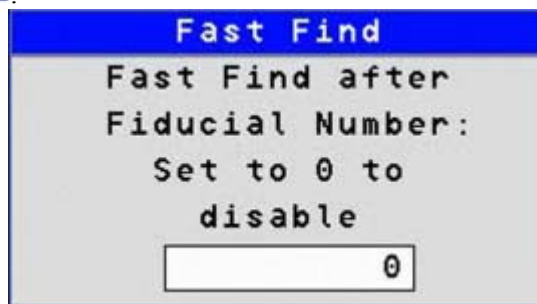
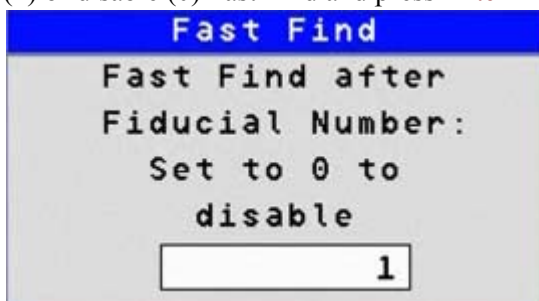
2. Press **X-axis Jog Arrows**  until Vision is highlighted in blue.

3. Press **Enter**  to access the Vision sub-menu.

4. Press **X-axis Jog Arrows**  until Fast Find is highlighted in blue.

5. Press **Enter**  to access the Fast Find function.

6. Enable (1) or disable (0) Fast Find and press **Enter** .



7. Press **Cancel**  to return to the Vision sub-menu. The status will be on the display.

Vision	
Focus Camera	
Fiducial Diam:	0.255
Correlation	: 0.200
Retake Dist	: 0.005
Settling Time:	100
Flip to Cut	: NO
Auto Focus	: NO
Fast Find	: 1

Vision	
Focus Camera	
Fiducial Diam:	0.250
Correlation	: 0.200
Retake Dist	: 0.000
Settling Time:	100
Flip to Cut	: NO
Auto Focus	: YES
Fast Find	: 0

[Vision](#)

[Menu](#)

Camera Delay

The Camera Delay function sets how long the system waits to move after taking a picture of the fiducial. If the value is too low, then the operator will see blurry images in Vision Tool. The Settling Time function applies before the picture is taken while the Camera Delay function applies after the picture is taken.

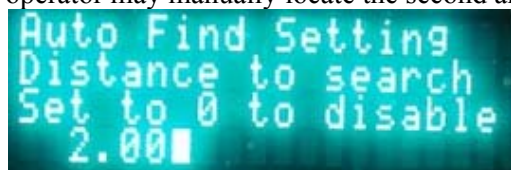
1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until Vision is highlighted in blue.
3. Press **Enter**  to access the Vision sub-menu.
4. Press **X-axis Jog Arrows**  until Camera Delay is highlighted in blue.
5. Press **Enter**  to access the camera delay.
6. Input a value for the delay (e.g., 100msec) and press **Enter** . The Vision sub-menu will show the amended value.

[Vision](#)[Menu](#)

Auto Find Setting

The Auto Find Setting function is used to specify the distance that the system will automatically scan for the second and later fiducials. After the operator manually finds the first fiducial, Auto Find Setting will search for the second and later fiducials by looking in half the field of view (FOV) area above, below, and next to the programmed location of the fiducials in the MultiVision job file.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**   until Vision is highlighted in blue.
3. Press **Enter**  to access the Vision sub-menu.
4. Press **X-axis Jog Arrows**   until Auto Find Setting is highlighted in blue.
5. Press **Enter**  to access the Auto Find Setting function.
6. Enter the distance for the camera to scan for the fiducial (e.g., 0.2" or 0.127 mm) or leave as "0" so the operator may manually locate the second and third fiducials.



7. Press **Enter**  to save the distance and return to the Vision sub-menu.

[Vision](#)[Menu](#)

Manual Find Setting

The Manual Find Setting function allows the operator to locate the fiducial with either the tool or the camera if the fiducial is not automatically found.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until Vision is highlighted in blue.
3. Press **Enter**  to access the Vision sub-menu.
4. Press **X-axis Jog Arrows**  until Manual Find Setting is highlighted in blue.
5. Press **Enter**  to access the Manual Find Setting function.
6. Press **X-axis Jog Arrows**  to move between “Jog Tool to Fid.” and “Jog Camera to Fid.” The double arrows or highlights show which setting is active.

7. Press **Enter**  and Jog the tool or camera (whichever was selected above) directly over the fiducial.
8. Press **Enter** . The camera will take a snapshot of the fiducial and continue locating fiducials based on their orientation from the manually found fiducial.

[Vision](#)[Menu](#)

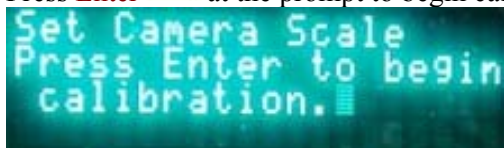
Set Camera Scale

The Set Camera Scale function calibrates the camera's colored pixels to a physical distance based upon the input diameter of the fiducial. The operator specifies a distance to which the machine will move, take a picture, move back, take a picture, calculate the scale, and save the camera scale. The camera scale can be set at the initial setup.

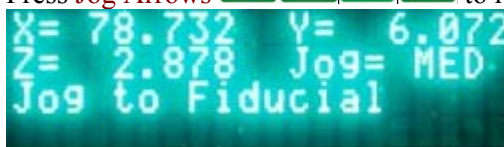
- 1 Press **Jog Arrows**  to reach a fiducial.
2. Press **Menu**  to access the menu system.
3. Press **X-axis Jog Arrows**  until Vision is highlighted in blue.
4. Press **Enter**  to access the Vision sub-menu.
5. Press **X-axis Jog Arrows**  until Set Camera Scale is highlighted in blue.
6. Press **Enter**  to access the Set Camera Scale function. The keypad display will prompt the operator for a move size. The operator specifies a distance to which the machine will move, take a picture, move back, take a picture, calculate the scale, and save the camera scale.
7. Enter a value of 0.2 for 1" FOV lens (0.127 for 6.35 mm) and press **Enter** . The camera will complete a series of 0.2" (0.127 mm) moves off the fiducial point as the camera checks for pixels per inch (mm). The scale will be based on these readings. A typical scale value will be 1000 ± 100 for a 1" (25.4mm) field of view.



8. Press **Enter**  at the prompt to begin calibration.



9. Press **Jog Arrows**  to reach the fiducial on the material to be cut and press **Enter** .



[Vision](#)

[Menu](#)

Set Camera Offsets

The Set Camera Offsets function is used to set the camera offsets based on the center of the spindle and the center of the camera by marking a hole with the Cut Depth entered in the Params_2D menu. The system prompts the operator to Jog over until the fiducial is close to the center of the computer screen. The location does not have to be exact. Camera offsets are usually set at installation and do not need to be reset unless the spindle or camera is replaced.

1. Spray paint a portion of the spoilboard or any scrap material and allow the paint to dry.



2. Set surface on the spray painted section using a 0.25" (6.35mm) End Mill.
3. Set Home in the spray painted section.
4. Set the Cut Depth from the Params_2D menu to cut into the spoilboard. Cut thicknesses may vary for each customer, though 0.01 – 0.02" (0.25 – 0.51mm) is typical.
 - a. Open Cut Depth from the Params_2D menu.



- b. Enter the appropriate thickness using the number keypad (e.g., 0.020" [0.51mm]).

- c. Press **Cancel**  to exit the Params_2D menu.

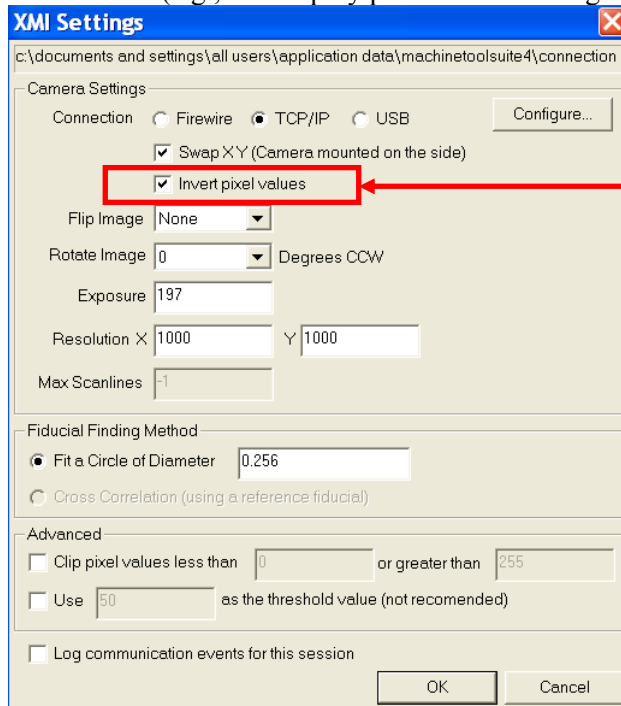
5. Set the Camera Offsets.
 - a. Open Setting Camera Offsets from the Vision menu.



- b. Press **Enter**  to allow the spindle to mark a fiducial at the current home. The spindle will turn on, and the router will plunge to the Cut Depth.



6. Open VisionTool check the box next to *Invert pixel values* when the contrast material is darker than the marked hole (e.g., black spray painted area and light material underneath as shown above).



7. Select OK.

8. Jog the camera until the fiducial is in the center of the VisionTool window on the computer screen.



9. Review the keypad display showing the offset amount and offset of fiducial and press **Enter** . This will save the offsets.



```
Camera Offset set to
X: -0.865   Y:  6.192
Fiducial Offsets
X:549.000   Y:623.000
```

The controller will save the camera offsets in Flash parameters **132** (X-axis offset) and **133** (Y-axis offset). The camera offsets will be saved in the computer until overwritten or deleted by the operator and do not need to be reset for each job.

10. Uncheck the box next to *Invert pixel values* and select *OK*.

[Vision](#)[Menu](#)

Move to Cam Offsets

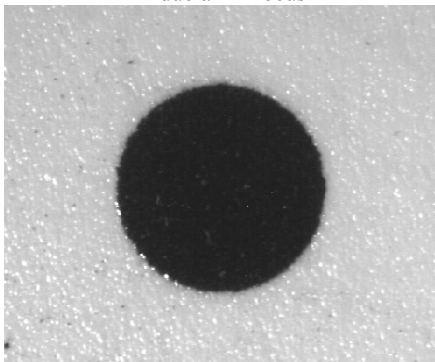
The Move to Camera Offsets function is mainly used as a verification tool to center the camera over the fiducial. Operators should only use this function once the Soft Home position is set over the marked fiducial.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until Vision is highlighted in blue.
3. Press **Enter**  to access the Vision sub-menu.
4. Press **X-axis Jog Arrows**  until Move to Cam Offsets is highlighted in blue.
5. Press **Enter**  to move the system to the camera offsets.

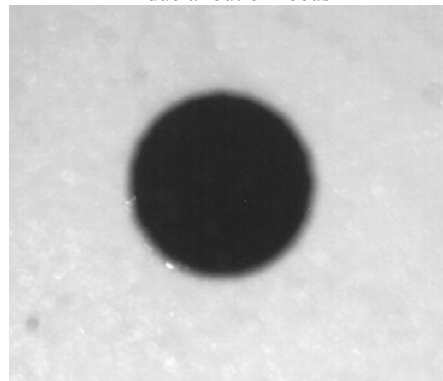


6. Click on Live Feed  in the toolbar.
7. Press **Jog Arrows**  to move the camera over a fiducial.
8. Press **Z-axis Jog Arrows**  to bring the fiducial into clearer view.

Fiducial in Focus



Fiducial out of Focus



9. Press **Enter**  once the fiducial is clear and return to the Vision sub-menu.

[Vision](#)

[Menu](#)

ATC

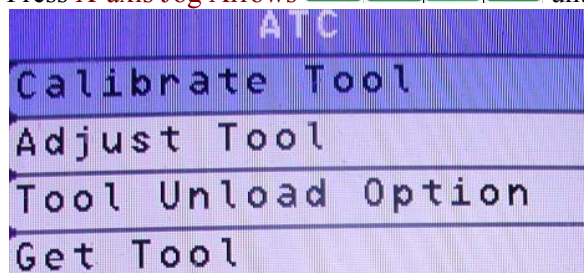
The Automatic Tool Changer (ATC) sub-menu is used to calibrate, adjust, load, and unload tools and is only available on machines with the ATC option.

1. Press **Menu**  to access the menu system.

2. Press **X-axis Jog Arrows**   until ATC is highlighted in blue.

3. Press **Enter**  to access the ATC sub-menu.

4. Press **X-axis Jog Arrows**     until the appropriate option is highlighted in blue.



Calibrate Tool	Adjust Tool
Tool Unload Option	Get Tool
LoadTools	CalGDrill
LoadDrill	AdjGDrill
CfgGDrill	

[Menu](#)

Calibrate Tool

The Calibrate Tool function is used to calibrate new tools when a new tool is inserted into the ATC. The machine will store the Z-axis tool offsets of the new tool, and this allows for fast tool changes. The Z-axis tool offset of each tool is stored in memory even after the machine is turned off.

1. Make sure the tool to be calibrated has been properly loaded in the ATC.

2. Press **Menu**  to access the menu system.

3. Press **X-axis Jog Arrows**  until ATC is highlighted in blue.

4. Press **Enter**  to access the ATC sub-menu.

5. Press **X-axis Jog Arrows**  until Calibrate Tool is highlighted in blue.

6. Press **Enter**  to access the Tool Calibration function.

7. Select the number of the tool to calibrate using the number keypad (e.g., 1).

8. Press **Enter** . The spindle will move over to the tool carousel, unload the tool if one is currently in place, rotate to the selected tool, and load that tool into the system. The system will then move over the calibration block.

9. Press the **Z-axis Jog Arrows**  to lower the spindle until it is near the top of the calibration block.

10. Place the grounding cradle on the ATC mounting groove in the spindle tool holder.

11. Press and hold **Zero**  to begin the tool calibration. Once **Zero**  is pressed, the Z-axis will move down slowly until it senses the calibration block. It will then lift off the block up to maximum Z height.

12. Remove the mounting collar before beginning work.

[ATC](#)[Menu](#)

Adjust Tool

The Adjust Tool function is used to adjust the length of a tool after the tool has been calibrated and the surface has been set. **Tools must be calibrated before they can be adjusted.**

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until ATC is highlighted in blue.
3. Press **Enter**  to access the ATC sub-menu.
4. Press **X-axis Jog Arrows**  until Adjust Tool is highlighted in blue.
5. Press **Enter**  to access the Adjust Tool function.
6. Enter the number of the tool to adjust using the number keypad (e.g., 1).



7. Press **Enter**  to adjust the tool.



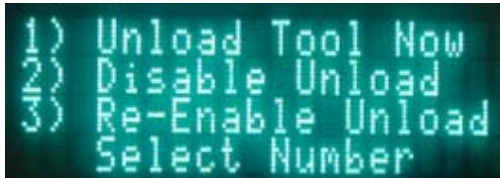
8. Enter the adjusting factor for the selected tool using the number keypad (e.g., -0.001). Positive numbers will increase and negative numbers will decrease the length of the tool.
9. Press **Enter**  to adjust the tool. The display will return to the ATC sub-menu.

[ATC](#)[Menu](#)

Tool Unload Option

The Tool Unload Option function allows the operator to choose if a tool will be unloaded automatically at the end of the job. This menu item is also available for operators to empty the spindle of any tool holder.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until ATC is highlighted in blue.
3. Press **Enter**  to access the ATC sub-menu.
4. Press **X-axis Jog Arrows**  until Tool Unload Option is highlighted in blue.
5. Press **Enter**  to unload a tool.
6. Enter the appropriate number using the number keypad to select the function to be performed (e.g., 2 will disable the automatic unload at the end of the job).



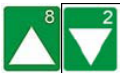
```
1) Unload Tool Now
2) Disable Unload
3) Re-Enable Unload
Select Number
```

7. Press **Enter**  to accept the selection. The system will return to the ATC sub-menu.

[ATC](#)[Menu](#)

Get Tool

The Get Tool function allows the operator to retrieve a specific tool and have it brought to the front of the machine.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until ATC is highlighted in blue.
3. Press **Enter**  to access the ATC sub-menu.
4. Press **X-axis Jog Arrows**  until Get Tool is highlighted in blue.
5. Press **Enter**  to access the Get Tool function and enter the tool number (e.g., 1).

6. Press **Enter**  to enable the spindle to retrieve the tool and bring it to the front of the machine.

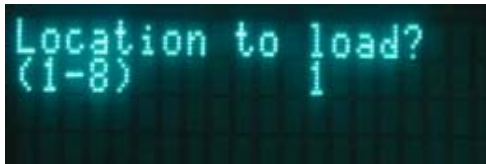
[ATC](#)[Menu](#)

Load Tools

The Load Tools function rotates the carousel to allow a new tool holder to be inserted. The current tool number will be displayed once LoadTools is selected. This option is only available on rotary ATC systems.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until ATC is highlighted in blue.
3. Press **Enter**  to access the ATC sub-menu.
4. Press **X-axis Jog Arrows**  until Load Tools is highlighted in blue.
5. Press **Enter**  to access the Load Tools function.

6. Enter the location number of the tool to be loaded (e.g., 1).



7. Press **Enter** . The carousel will rotate to that tool number location.
8. Load the tool into the corresponding slot for that number.
9. Repeat steps 6-8 for the remaining tools.
10. Press **Cancel**  to rotate the carousel back to its original position.

[ATC](#)[Menu](#)

Calibrate Gang Drill

The Calibrate Gang Drill function calibrates new drill bits that are inserted into the Gang Drill and stores the Z-axis tool offsets of new drill bits in the controller memory. Gang Drill Calibration can only be completed after the operator has loaded drills.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until ATC is highlighted in blue.
3. Press **Enter**  to access the ATC sub-menu.
4. Press **X-axis Jog Arrows**  until CalGDrill is highlighted in blue.
5. Press **Enter**  to calibrate a gang drill. The #1 drill will move over to the calibration block or surface block.
6. Press **Z-axis Jog Arrows**  to lower the spindle until it is near the top of the block.



 - a. Press **One**  for “Manual” when calibrating only 1 drill after a bit change. When the drill has been calibrated, the display will return to the ATC sub-menu.
 - b. Press **Two**  for “Auto” to begin the tool calibration. The Z-axis will move down slowly until it senses the block and will then lift off the block up to maximum Z height. Each drill will move over the block and go through the same motion until all are calibrated. When all of the drills are calibrated, the display will return to the ATC sub-menu.

[ATC](#)[Menu](#)

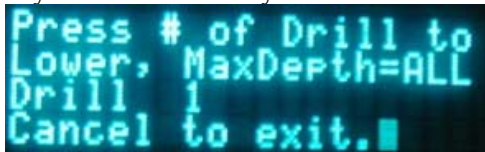
Load Drill

The Load Drill function fires a specific drill to be manually unloaded or loaded.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until ATC is highlighted in blue.
3. Press **Enter**  to access the ATC sub-menu.
4. Press **X-axis Jog Arrows**  until Load Drill is highlighted in blue.
5. Press **Enter**  to load a drill. The machine will move to the Hard Home position.
6. Enter the number of the drill to be lowered (e.g., 1) or press **Set Max Depth**  to lower all drills.



7. Press numbers 2 through 9 to access each following drill. Any currently lowered drill is raised before the selected drill is lowered, and the lowered drill is displayed on the keypad. Operators can press **Zero** to retract any drill that is already lowered.



8. Adjust the set screw so that any existing bit can be removed or a new bit can be inserted.
9. Insert a drill bit into the drill.
10. Adjust the set screw to hold the bit securely.
11. Repeat steps 7-10 for each drill as needed.
12. Press **Cancel**  to retract any lowered drill and return the display to the ATC sub-menu.

[ATC](#)[Menu](#)

AdjGDrill

The Adjust Gang Drill function allows the operator to adjust the length of a drill after the drill has been calibrated and the surface has been set. This prevents the operator from having to redo the settings on all drills when only one drill needs to be modified.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**   until ATC is highlighted in blue.
3. Press **Enter**  to access the ATC sub-menu.
4. Press **X-axis Jog Arrows**   until AdjGDrill is highlighted in blue.
5. Press **Enter**  to access the gang drill settings individually.
6. Enter the number of the drill to be adjusted (e.g., 1) and press **Enter** .



7. Press **X-axis Jog Arrows**   to adjust the drill by the thousandths. Operators can make positive or negative adjustments.



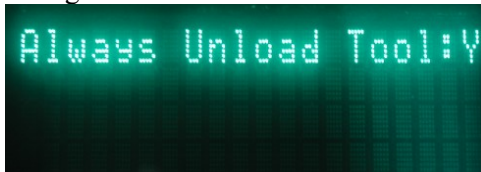
8. Press **Enter**  to save the adjustment and return to the ATC sub-menu.

[ATC](#)[Menu](#)

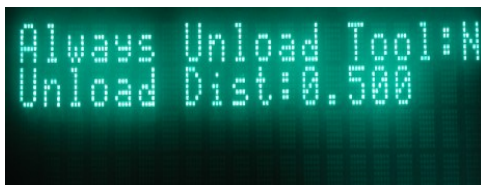
CfgGDrill

The Configure Gang Drill function sets the system to always put away the tool if the gang drill is called out in the job file.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until the blinking cursor is over “A” in ATC.
3. Press **Enter**  to access the ATC sub-menu.
4. Press **X-axis Jog Arrows**  until the blinking cursor is over the “C” in CfgGDrill.
5. Press **Enter**  to access the gang drill configuration.
6. Press **Enter**  to toggle between Yes (Y) and No (N). Operators can set the unload distance if the configuration is set to No.



Always Unload Tool:Y



Always Unload Tool:N
Unload Dist:0.500

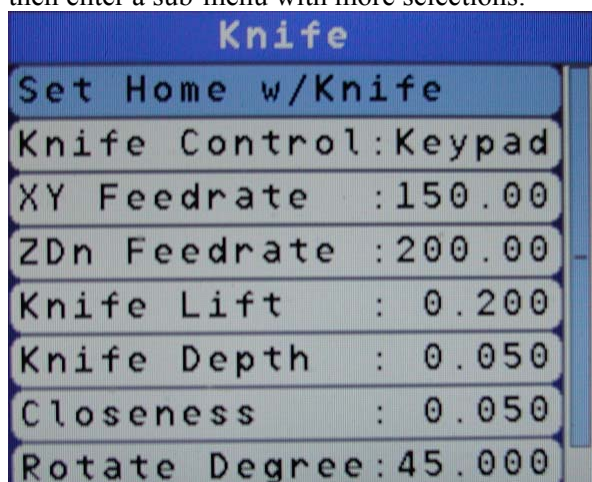
7. Press **Cancel**  to return to the ATC sub-menu.

[ATC](#)[Menu](#)

Knife

The Knife sub-menu is used to establish the knife parameters specific to a routing machine. The cartridge knife offers up to 20 options for blade types. Most operators using the knife option are cutting vinyl, cloth, or plastic.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**   until Knife is highlighted in blue.
3. Press **Enter**  to access the Knife sub-menu.
4. Press **X-axis Jog Arrows**   until the appropriate option is highlighted in blue. Certain menu items will then enter a sub-menu with more selections.

[Set Home w/Knife](#)[XY Feedrate](#)[Knife Lift](#)[Closeness](#)[Knife Dn Dly](#)[Min. Radius](#)[Knife Control](#)[ZDn Feedrate](#)[Knife Depth](#)[Rotate Degree](#)[Knife Up Dly](#)[Menu](#)

Set Home w/Knife

The Set Home w/Knife function allows a Soft Home to be set using the center line of the knife. This item is mainly used in the cartridge knife system when operators are not including the spindle in the job file, and the setting can be turned ON and OFF.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**   until Knife is highlighted in blue.
3. Press **Enter**  to access the Knife sub-menu.
4. Press **X-axis Jog Arrows**   until Set Home w/Knife is highlighted in blue.
5. Press **Enter**  to set Home with the Knife.
6. Press **Jog Arrows**     until the knife is at the Home location and press **Enter** .



X= 3.357 Y= -9.438
Z= -8.726 Jog= FAST
Set Home w/Knife 1
Enter=Set

[Knife](#)[Menu](#)

Knife Control

The Knife Control function specifies whether the Knife Lift and Knife Depth will be set by the job file or at the keypad. XY Feedrate, ZDn Feedrate, Knife Lift, and Knife Depth menu items are only available in the Knife sub-menu when the Knife Control is set to Keypad.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**   until Knife is highlighted in blue.
3. Press **Enter**  to access the Knife sub-menu.
4. Press **Enter**  to toggle between Job and Keypad.



```
Set Home w/Knife 1
Knife Control:Job
XY Feedrate : 20.00
ZDn Feedrate : 200.0
```



```
Set Home w/Knife 1
Knife Control:Keypad
XY Feedrate : 20.00
ZDn Feedrate : 200.0
```

[Knife](#)[Menu](#)

XY Feedrate

The Knife Feedrate function sets the cartridge knife cutting speed during the cutting sequence. This item is only available if Knife Control is set to Keypad.

1. Press **Menu**  to access the menu system.

2. Press **X-axis Jog Arrows**   until Knife is highlighted in blue.

3. Press **Enter**  to access the Knife sub-menu.

4. Press **X-axis Jog Arrows**   until XY Feedrate is highlighted in blue.

5. Press **Enter**  to set the cutting speed.



Cut Speed? 20.0
Currently 80.0%
Enter X,Y Feedrate

6. Input the cutting speed (e.g., 25) and press **Enter** . The value will be displayed on the Knife sub-menu.



Set Home w/Knife 1
Knife Control: Keypad
XY Feedrate : 25.00
ZDn Feedrate : 200.0

[Knife](#)

[Menu](#)

ZDn Feedrate

The ZDn Feedrate function sets how fast the blade enters the material. This item is only available if Knife Control is set to Keypad and is only applicable if the cartridge configuration is set to Z controlled.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**   until Knife is highlighted in blue.
3. Press **Enter**  to access the Knife sub-menu.
4. Press **X-axis Jog Arrows**   until ZDn Feedrate is highlighted in blue.
5. Press **Enter**  to set the Z-axis speed.
6. Input the Z-axis speed (e.g., 250) and press **Enter** . The value will be displayed on the Knife sub-menu.

[Knife](#)[Menu](#)

Knife Lift

The Knife Lift function moves the knife above the material surface by a specified distance to rotate to a new degree and/or move to the next cut contour. With Cartridge knives, the Knife Lift option is only available if Knife Control is set to Keypad; otherwise, the Z values from the job file are used.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**   until Knife is highlighted in blue.
3. Press **Enter**  to access the Knife sub-menu.
4. Press **X-axis Jog Arrows**   until Knife Lift is highlighted in blue.
5. Press **Enter**  to set the Knife Lift.



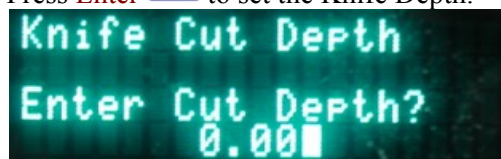
6. Enter lift height (e.g., 1.500) and press **Enter** . The value will be displayed on the Knife sub-menu.

[Knife](#)[Menu](#)

Knife Depth

The Knife Depth function refers to the Z screw distance that the knife will move down from the surface of the material during the file execution. With Cartridge knives, the Knife Depth menu option is only available if Knife Control is set to Keypad; otherwise, the Z values from the job file are used.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**   until Knife is highlighted in blue.
3. Press **Enter**  to access the Knife sub-menu.
4. Press **X-axis Jog Arrows**   until Knife Depth is highlighted in blue.
5. Press **Enter**  to set the Knife Depth.



6. Enter a cut depth (e.g., 0.0) and press **Enter** . The value will be displayed on the Knife sub-menu.

[Knife](#)[Menu](#)

Closeness

The Closeness function is used to set how closely the knife will follow the job file. The Closeness value is calculated from the vertex point inward between the two straight lines. Although the input value is the same for all the varying degrees (e.g., 0.010 closeness on angles of 22°, 95°, and 154°), it affects those angles differently. The more obtuse or dull the angle, the more effect the closeness value of 0.010 has. The more acute or sharp the angle, the less effect the closeness value has.

- Operators who wish to **enable rounding** should set a low closeness value and a high minimum radius value.
- Operators who wish to **disable rounding** should set a high closeness value and a low minimum radius value.

If the arc from the closeness value is less than the minimum radius value, then the knife will move to the vertex, lift up to the Knife Lift Height, rotate to follow the path exactly, and re-enter the material to continue the job file.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**   until Knife is highlighted in blue.
3. Press **Enter**  to access the Knife sub-menu.
4. Press **X-axis Jog Arrows**   until Closeness is highlighted in blue.
5. Press **Enter**  to adjust the Closeness value.
6. Enter the closeness value using the number keypad (e.g., 0.010).



7. Press **Enter**  to save the value and return to the Knife sub-menu where the Closeness value will be displayed.

[Knife](#)[Menu](#)

Rotate Degree

The Rotate Degree function sets the degree the blade can stay down in the material and rotate if the blade is at the vertex point of an angle. **Rotate Degree is only used when the closeness arc created is smaller than the minimum radius**; therefore, this parameter is only used when the closeness value is very low (e.g., 0.010) and the minimum radius is very high (e.g., 0.750).

The controller will move the knife to the vertex point and then compare the relative angle of the required theta motion to the rotate degree value input at the keypad. With the knife tip at the vertex angle, the knife will twist to the new angle while still in the material and then continue in X & Y. Some operators will use a lower degree for thicker materials and a higher degree for thinner materials to prevent extra indentations in the material from where the knife rotated.

- Operators who wish to **enable rounding** should set a low closeness value, a high minimum radius value, and a rotate degree value of greater than 90.
- Operators who wish to **disable rounding** should set a high closeness value, a low minimum radius value, and a rotate degree value of 90.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until Knife is highlighted in blue.
3. Press **Enter**  to access the Knife sub-menu.
4. Press **X-axis Jog Arrows**  until Rotate Degree is highlighted in blue.
5. Press **Enter**  to adjust the rotation degree.
6. Enter the maximum rotation degree using the number keypad (e.g., 15.000).



7. Press **Enter**  to save the value and return to the Knife sub-menu where the Rotate Degree value will be displayed.

[Knife](#)[Menu](#)

Knife Dn Delay

The Knife Down Delay function is only used when a cartridge is configured as Solenoid controlled. When the cartridge is lowered, the Knife Down Delay allows the system to wait for the cartridge to lower into position. If the cartridge is Z controlled, then no delay will occur.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**   until Knife is highlighted in blue.
3. Press **Enter**  to access the Knife sub-menu.
4. Press **X-axis Jog Arrows**   until Knife Dn Delay is highlighted in blue.
5. Press **Enter**  to set the Knife Dn Delay.
6. Enter the value (e.g., 1000msec) and press **Enter** . The value will be displayed on the Knife sub-menu.

[Knife](#)[Menu](#)

Knife Up Delay

The Knife Up Delay function is only used when a cartridge is configured as Solenoid controlled. When the cartridge is raised, the Knife Up Delay allows the system to wait for the cartridge to raise into position. If the cartridge is Z controlled, then no delay will occur.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until Knife is highlighted in blue.
3. Press **Enter**  to access the Knife sub-menu.
4. Press **X-axis Jog Arrows**  until Knife Up Delay is highlighted in blue.
5. Press **Enter**  to set the Knife Up Delay.
6. Enter the value (e.g., 1000msec) and press **Enter** . The value will be displayed on the Knife sub-menu.

[Knife](#)[Menu](#)

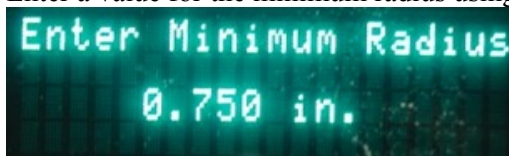
Min. Radius

The Minimum Radius function establishes the smallest radius that the knife will cut and is only applicable at 2 connecting line entities. If the arc created by the closeness value is larger than the minimum radius value, then the knife will follow the closeness value in the X- and Y-axes. This effectively keeps the knife in the material for a smooth motion.

If the arc that is automatically created by the closeness value is smaller than the minimum radius value, then the knife will move to the vertex point, lift, rotate, lower back down, and continue cutting. The parameters at the keypad have little or no effect on arc commands in G Code.

- Operators who wish to **enable rounding** should set a high minimum radius value and a low closeness value.
- Operators who wish to **disable rounding** should set a low minimum radius value and a high closeness value.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until Knife is highlighted in blue.
3. Press **Enter**  to access the Knife sub-menu.
4. Press **X-axis Jog Arrows**  until Min. Radius is highlighted in blue.
5. Press **Enter**  to access the minimum radius setting.
6. Enter a value for the minimum radius using the number keypad (e.g., 0.750).



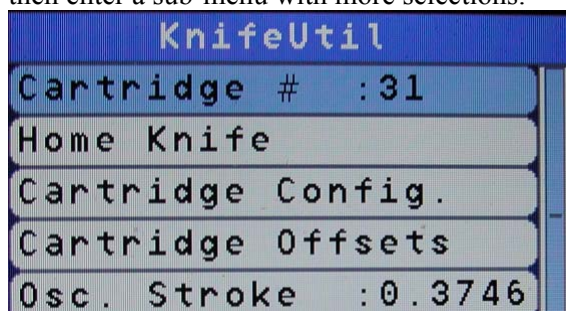
7. Press **Enter**  to save the value and return to the Knife menu where the Minimum Radius value will be displayed.

[Knife](#)[Menu](#)

KnifeUtil

The Knife Utility sub-menu sets parameters specific to the cartridge knife.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until KnifeUtil is highlighted in blue.
3. Press **Enter**  to access the Knife Utility sub-menu.
4. Press **X-axis Jog Arrows**  until the appropriate option is highlighted in blue. Certain menu items will then enter a sub-menu with more selections.



[Cartridge Number](#)

[Cartridge Config.](#)

[Osc Stroke](#)

[Knife Osc Test](#)

[Set Knife 2 Res](#)

[Home Knife](#)

[Cartridge Offsets](#)

[Move to Knife Offset](#)

[Set Knife 1 Res](#)

[Menu](#)

Cartridge Number

The Cartridge # function rotates the cartridge receiver to that cartridge number's offset for visual verification. The system will default to Cartridge 31 at initial boot-up and also during regular operations if no cartridge is already set up in the machine.

1. Press **Menu**  to access the menu system.

2. Press **X-axis Jog Arrows**   until KnifeUtil is highlighted in blue.

3. Press **Enter**  to access the Knife Utility sub-menu.

4. Press **X-axis Jog Arrows**   until Cartridge # is highlighted in blue.

5. Press **Enter**  to access the cartridge.



6. Input the cartridge number and press **Enter** . The display returns to the KnifeUtil sub-menu showing the selected cartridge.

[KnifeUtil](#)

[Menu](#)

Home Knife

The Home Knife function initiates the Homing macro for the knife in case the stepper motor loses its physical location. Operators will not have to re-home the knife unless the Flash parameter changes or the knife has lost the last physical Home location.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**   until KnifeUtil is highlighted in blue.
3. Press **Enter**  to access the KnifeUtil sub-menu.
4. Press **X-axis Jog Arrows**   until Home Knife is highlighted in blue.
5. Press **Enter**  to access the Home Knife function.



6. Wait until the Knife has located its Home position and the keypad shows the KnifeUtil sub-menu before continuing with the setup or cut sequence.

[KnifeUtil](#)[Menu](#)

Cartridge Config

The Cartridge Config function determines how each knife operates in a job. Each knife can be configured as drag or oscillating, and the oscillations will become active after the knife is selected. Additionally, each knife can be Z controlled, Solenoid controlled, or Angled. Operators **must** resurface and recalibrate the cartridge knife after changing configurations.

- If Z controlled, the Cartridge Configuration function uses the Knife Depth and Lift values from the executed M- & G-Code file. Z Control is the preferred method for cutting material that is 1" (25.4mm) or thicker.
- If Solenoid controlled, the Cartridge Configuration function uses the Knife Up Dly and Knife Dn Dly in addition to raising and lowering the knife. Solenoid controlled is the preferred method for cutting material that is 3/4" (19.1mm) or thinner.
- If Angled, the Cartridge Configuration function accounts for the 45° angle of the blade

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until KnifeUtil is highlighted in blue.

3. Press **Enter**  to access the Knife Utility sub-menu.

4. Press **X-axis Jog Arrows**  until Cartridge Config is highlighted in blue.

5. Press **Enter**  to access the cartridge configuration list.

```
Cart.31 :Osci-Z
Cart.32 :Drag-Sol
Cart.33 :Drag-Sol
Cart.34 :Drag-Sol
```

6. Press **X-axis Jog Arrows**  to reach the applicable cartridge number and press **Enter** .

```
Cartridge 31
> *Oscill Drag
  *Zup/dn Sol
  Angle *Straight
```

7. Press **X-axis Jog Arrows**   to reach the appropriate parameter and **Y-axis Jog Arrows**   to change the setting. The asterisk (*) indicates which setting is active.

If the cartridge is configured as Solenoid rather than Z up/dn, then the Angle/Straight option is no longer available.



8. Press **Enter**  to return to the Cartridge Configuration sub-menu or **Cancel**  to return to the KnifeUtil sub-menu.

[KnifeUtil](#)

[Menu](#)

Cartridge Offsets

The Cartridge Offsets function is used at the initial setup to set the orientation of the knife parallel to the X-axis. The sharp edge of the blade should point to the back of the machine. Cartridge 31 is the limit offset for knife head 1 after homing, and cartridge 41 is the limit offset for knife head 2 after homing. The cartridge offsets 32 – 40 are relative to cartridge offset 31.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until KnifeUtil is highlighted in blue.
3. Press **Enter**  to access the Knife Utility sub-menu.
4. Press **X-axis Jog Arrows**  until Cartridge Offsets is highlighted in blue.
5. Press **Enter**  to access the Cartridge Offsets.

Cartridge Offsets		
Cart. 31	Off:	0.274
Cart. 32	Off:	0.000
Cart. 33	Off:	0.000
Cart. 41	Off:	0.018
Cart. 42	Off:	0.036
Cart. 43	Off:	0.000

6. Press **Z-axis Jog Arrows**  to reach the specific cartridge and press **Enter** . The machine will find the knife home and then issue the following prompt:

```

K Axis          0.000
Cart:  31      Jog= MED
Jog to Knife Offset
Press ENTER
  
```

7. Press **Jog Arrows**  to Jog to the knife offset and press **Enter** . The display will return to the Cartridge Offsets sub-menu.

[KnifeUtil](#)

[Menu](#)

Osc Stroke

The Osc Stroke function shows the value that will be added to the length of the tool during the surfacing or calibrating of the knife when the knife is not oscillating. This setting is only valid on an oscillating cartridge with the oscillations turned off during setting surface or calibrating. The variable stroke is based on the PSI set by the regulator.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until KnifeUtil is highlighted in blue.
3. Press **Enter**  to access the Knife Utility sub-menu.
4. Press **X-axis Jog Arrows**  until Osc Stroke is highlighted in blue.
5. Press **Enter**  to set the value of the oscillation stroke.



6. Input a value (e.g., 0.340) and press **Enter** . The display will return to the KnifeUtil sub-menu where the Oscillating Stroke will be displayed.

[KnifeUtil](#)

[Menu](#)

Move to Knife Offset

The Move to Knife Offset function validates the correct setting for the cartridge or knife offsets from the center of the spindle.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until KnifeUtil is highlighted in blue.
3. Press **Enter**  to access the Knife Utility sub-menu.
4. Press **X-axis Jog Arrows**  until Move to Knife Offset is highlighted in blue.
5. Press **Enter**  to access the Move to Knife Offset function.
6. Enter the knife number (e.g., 1 or 2).



7. Press **Enter**  to move to the offset. The display will return to the KnifeUtil sub-menu.

[KnifeUtil](#)[Menu](#)

Knife Osc Test

The Knife Oscillator Test checks the Knife Oscillator output.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**   until KnifeUtil is highlighted in blue.
3. Press **Enter**  to access the Knife Utility sub-menu.
4. Press **X-axis Jog Arrows**   until Knife Osc Test is highlighted in blue.
5. Press **Enter**  to access the oscillation test.

6. Enter the knife number (e.g., 1) and press **Enter** .

```
Enter Knife Number?  
( 1 - 2 )  
  
1
```

7. Press **One**  to toggle the oscillation test ON and OFF.

```
X = - 0 . 3 6 1      Y = - 5 . 2 9 1  
Z =  0 . 0 0 0      Jog = MED  
1 = Knife Osc on/off  
Osc1(off)
```

```
X = - 0 . 3 6 1      Y = - 5 . 2 9 1  
Z =  0 . 0 0 0      Jog = MED  
1 = Knife Osc on/off  
Osc1(on)
```

8. Press **Cancel**  to exit the oscillation test and return to the Knife Utility sub-menu.

[KnifeUtil](#)

[Menu](#)

Set Knife 1 Res

The Set Knife 1 Resolution function defines the number of pulses needed to move the cartridge exactly 360°. This setting will not have to be readjusted unless the knife is replaced.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**   until KnifeUtil is highlighted in blue.
3. Press **Enter**  to access the Knife Utility sub-menu.
4. Press **X-axis Jog Arrows**   until Set Knife 1 Res is highlighted in blue.



Set Knife Resolution?
ENTER=Yes, EXIT=No

5. Press **Enter**  to access the Set Knife 1 Resolution function.



Knife Head 1 Offset
X: 3.515
Y: -9.438
Press Enter to Move

6. Press **Enter**  to set the resolution.



Enter number of Revs

7. Enter the number of revolutions and press **Enter** . The knife will complete the input number of revolutions, count down on the display, and display any changes to Knife 1 resolution.



Changing Knife Res
From -30000.
to -30001.

8. Press **Enter**  to save the new resolution and return to the KnifeUtil sub-menu.

[KnifeUtil](#)[Menu](#)

Set Knife 2 Res

The Set Knife 2 Res defines the number of pulses needed to move the cartridge exactly 360°. This setting will not have to be readjusted unless the knife is replaced. This option is not available on single-knife systems.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until KnifeUtil is highlighted in blue.
3. Press **Enter**  to access the Knife Utility sub-menu.
4. Press **X-axis Jog Arrows**  until Set Knife 2 Res is highlighted in blue.



Set Knife Resolution?
ENTER=Yes, EXIT=No

5. Press **Enter**  to access the Set Knife 2 Resolution function.



Knife Head 1 Offset
X: 3.515
Y: -9.438
Press Enter to Move

6. Press **Enter**  to set the resolution.



Enter number of Revs

7. Enter the number of revolutions and press **Enter** . The knife will complete the input number of revolutions, count down on the display, and display any changes to Knife 2 resolution.



Changing Knife Res
From -30000.
to -30001.

8. Press **Enter**  to save the new resolution and return to the KnifeUtil sub-menu.

[KnifeUtil](#)[Menu](#)

Material Handling

The Material Handling sub-menu is used to operate the material handling and material pushing option on the 5000 Series XD line of machines.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until MatlHand is highlighted in blue.
3. Press **Enter**  to access the Material Handling sub-menu.
4. Press **X-axis Jog Arrows**  until the appropriate option is highlighted in blue. Certain menu items will then enter a sub-menu with more selections.

[Roller Setup ...](#)[Material Pusher ...](#)[Material Lifter ...](#)[Air/Knife Sweeper](#)[PopUp Pins](#)[Menu](#)

Roller Setup

The Roller Setup sub-menu is used to operate the rollers used in holding down multiple sheets of material. The rollers can be operated in sequence and can be tested individually.

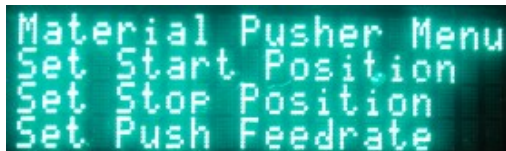
1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until MatlHand is highlighted in blue.
3. Press **Enter**  to access the Material Handling sub-menu.
4. Press **X-axis Jog Arrows**  until Roller Setup is highlighted in blue.
5. Press **Enter**  to access the Roller Setup.
6. Press **X-axis Jog Arrows**  to select the appropriate option.
 - Work Start will set the coordinate location for the initial cutting sequence.
 - Sheet Size will configure the system for the size of the material.
 - Enable Rollers will allow the rollers to be activated once the material is in the proper place.
 - Pre-Lift will allow a certain amount of space before lifting the material.
 - Roller 1 Position will show the coordinate location of roller 1 relative to the spindle.
 - Roller 2 Position will show the coordinate location of roller 2 relative to the spindle.
 - Roller 3 Position will show the coordinate location of roller 3 relative to the spindle.
 - Roller 4 Position will show the coordinate location of roller 4 relative to the spindle.
 - Test I/O will complete the internal connection to determine communication.
7. Press **Cancel**  to return to the Material Handling sub-menu.

[Material](#)[Menu](#)

Material Pusher

The Material Pusher sub-menu is used to move the material into place for cutting and off the table once the cutting sequence has been completed. For safety reasons, the Material Pusher will only fire down over the work and unload areas of the table

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until MatlHand is highlighted in blue.
3. Press **Enter**  to access the Material Handling sub-menu.
4. Press **X-axis Jog Arrows**  until Material Pusher is highlighted in blue.
5. Press **Enter**  to access the Material Pusher sub-menu.



6. Press **X-axis Jog Arrows**  to select the appropriate option.
 - Set Start Position will set the beginning position for the material pusher.
 - Set Stop Position will set the ending position for the material pusher.
 - Set Push Feedrate will set the speed at which the pusher will operate.
 - Auto Unload will unload the material automatically.
 - Test Cycle will complete a sequence to determine response.
 - Test I/O will complete the internal connection to determine communication.

7. Press **Cancel**  to return to the Material Handling sub-menu.

[Material](#)[Menu](#)

Material Lifter

The Material Lifter sub-menu is used to create suction on the material, lift the material from the stack, and raise the material to the proper height before lowering it to the cutting table.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until MatlHand is highlighted in blue.
3. Press **Enter**  to access the Material Handling sub-menu.
4. Press **X-axis Jog Arrows**  until Material Lifter is highlighted in blue.
5. Press **Enter**  to access the Material Lifter sub-menu.
6. Press **X-axis Jog Arrows**  to select the appropriate option.
 - Num Sheets will indicate the number of sheets of material with a maximum of 5 sheets.
 - Auto Load will set the configuration to load at the start of a job or at the sheet.
 - Load Prompt will set the configuration to prompt at the start of a job or at the sheet.
 - Lift Delay will set the time lapse before the material lifter will raise.
 - X Pickup will specify the X-axis location at the back to pick up the material.
 - X Drop-off will specify the X-axis location to drop off the material for final adjustments.
 - Load Speed will set the speed at which the material should be loaded.
 - XPos Dist will indicate the X position for laying the material onto the table.
 - XPos Speed will indicate the feedrate at which the material will be moved onto the table.
 - Test Cycle will complete a sequence to determine response.
 - Put Sheet Away will place the sheet of material out of the cutting area.
 - Test I/O will complete the internal connection to determine communication.
7. Press **Cancel**  to return to the Material Handling sub-menu.

[Material](#)[Menu](#)

Air/Knife Sweeper

The Air/Knife Sweeper sub-menu is used to test the responsiveness of the air knife by blowing a high-pressure airflow onto the table between the front roller and the sweeper.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**   until MatlHand is highlighted in blue.
3. Press **Enter**  to access the Material Handling sub-menu.
4. Press **X-axis Jog Arrows**   until Air/Knife Sweeper is highlighted in blue.
5. Press **Enter**  to access the Air/Knife Sweeper sub-menu.



6. Press **X-axis Jog Arrows**   to select the appropriate option.
 - Sweeper Delay will impose a set period of time in microseconds before operating the air knife sweeper.
 - Test I/O will complete the internal connection to determine communication.
7. Press **Cancel**  to return to the Material Handling sub-menu.

[Material](#)[Menu](#)

PopUp Pins

The PopUp Pins sub-menu is used to test the responsiveness of the popup pins and the level of connectivity.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until MatlHand is highlighted in blue.
3. Press **Enter**  to access the Material Handling sub-menu.
4. Press **X-axis Jog Arrows**  until PopUp Pins is highlighted in blue.
5. Press **Enter**  to access the PopUp Pins sub-menu.



6. Press **X-axis Jog Arrows**  to select the appropriate option.
 - Test I/O will complete the internal connection to determine communication.
7. Press **Cancel**  to return to the Material Handling sub-menu.

[Material](#)

[Menu](#)

Auto / All

The Auto / All function is used to set the cutting sequence to apply to both heads or to just one head at a time. This menu item is only available on standard routers with multiple cutting heads.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until Auto/All is highlighted in blue.
3. Press **Enter**  to access the Auto/All function.
4. Press **Enter**  to toggle between Auto and All.
7. Press **Cancel**  to return to the main menu.

[Menu](#)

Z Sync

The Z Sync function is used to transfer all head 1 Z settings over to head 2. This menu item is only available on standard routers with multiple cutting heads.

1. Press **Menu**  to access the menu system.
2. Press **X-axis Jog Arrows**  until Z Sync is highlighted in blue.
3. Press **Enter**  to access the Z Sync function.
4. Press **Enter**  to toggle between Enabled and Disabled.
7. Press **Cancel**  to return to the main menu.

[Menu](#)

Options

[top](#)

Digitize	2
Digitize Commands	2
Manual Digitize	4
Rapid Shape	5
Using Rapid Shape	6
Rapid Shape Tips and Troubleshooting	8
Laser Pointer	9
Bar Code Scanner	10
Chip Removal Options	11
Pressure Foot	11
Dust Collection Hoods	12
Dust Collection Hoods Installation for the Dust Boot	14
Engraver	15
Pneumatic Drill	15
Vinyl Knife	16
Under Gantry Pusher	16
Heat Exchanger	17
Cold Air Gun	17
Table Format	18
Vacuum Options	19
Vacuum Zones	20
Locator Pins	21
Safety Mat	22
Light Curtains	23
Coolubricator System	24
Operating the Coolubricator System	25
Troubleshooting the Coolubricator System	26

Digitize

The Digitize function is used for the **MultiCam** Rapid Shape feature or Manual Digitizing. With either feature, the operator has the ability to duplicate simple and complex shapes on the router. This will record the outline on the host PC. Rapid Shape and Manual Digitizing both create a G-Code file that can be imported into many CAD/CAM systems. **MultiCam** does offer a DXF conversion file that will allow the G-Code file to be imported into many design software packages.



Before starting Digitize, the operator must verify that JobNameServer is visible in the Windows system tray, the small box in the bottom corner by the system clock, as this means that JobNameServer is active and running on the computer. If this icon is displayed, the operator is ready to digitize the machine. Not all **MultiCam** Routers carry the Digitize option, but the Digitize option can be purchased from the local distributor or from **MultiCam**. [Options](#)

Digitize Commands

The Digitize Commands are utilized in Manual Digitizing and Rapid Shape, and only a few of these commands are used in Rapid Shape.



1. **Linear** (**Feedrate Increase**) allows straight line, point-to-point moves in these G-Code Commands: G00 for PU, G01 for PD.



2. **Spline** (**Feedrate Increase**) allows moving a series of points that make up a spline in these G-Code Commands: G25, G26, G27.
 - a. Press the Spline key to start the spline command.
 - b. Enter as many points as needed around the curve.
 - c. Press the Spline key to end the spline command and return to Linear mode.



3. **Arc** (**Feedrate Decrease**) allows an arc to be specified in these G-Code Commands: G02 for clockwise, G03 for counter-clockwise. The Arc will be calculated and presented. After establishing the 3 points of the Arc, the system will return to Linear mode.
 - a. Enter 3 points: First, Middle, and End.
 - b. Press the Arc key to start the arc command.
 - c. Jog to the first point and press **Enter**
 - d. Jog to the midpoint and press **Enter**
 - e. Jog to the end point and press **Enter**

4. **Back Up**  (**Spindle RPM Decrease**) moves the machine back up to the last Pen Down (PD) point entered. Typically 4 points are saved in the point buffer, which only saves PD moves, to allow for backing up and for generating corners.
5. **Set Home**  (**Set Home**) sets the Soft Home to the current position. The current point will become 0,0.
6. **Go Home**  (**Go Home**) moves to the current Soft Home position.
7. **Close Contour**  (**Drive Enable {3}**) moves to the start of the last PD. A contour is defined as a complete PD to Pen Up (PU) sequence.
8. **2D/3D**  (**Move {7}**) toggles between 2D and 3D: 2D sets X and Y while 3D sets X, Y, and Z. Currently, only 2D is available.
9. **PD/PU**  (**Set Max Depth**) toggles between PD and PU mode in these G-Code Commands: M11 for 2D PD, M21 for 2D PU, M12 for 3D PD, M22 for 3D PU. A point will not be issued at this time as this command only toggles the state. The PD or PU command will be generated on the following move. This allows the operator to produce only one command even if the PD/PU key is pressed multiple times.
10. **Exit**  (**Cancel/Exit**) ends the Job and exits Digitizing in the G-Code Command M02. If the current PD/PU state is PD, then an M21 will be issued to lift the pen prior to exiting.
11. **Enter**  (**Enter**) enters a point. When entering a point in Linear mode, either a G00 or a G01 will be output. The status of the PD or PU will determine which command is generated. A G27 will be output in Spline mode.
12. **Flush Buffer**  (**Spindle RPM Increase**) outputs the current point buffer to the Host PC. When setting points, the operator can save several points in a point buffer prior to being sent so they are not immediately visible in the Digitize Window. The Flush Buffer key will flush this buffer, outputting all points in the buffer so they can be viewed in the Digitize Window. It is not necessary to flush the buffer as the system will automatically do this at the end of the job.

[Options](#)

Manual Digitize

The Manual Digitize function creates a G-Code file one line at a time. When manually digitizing, operators must be sure to enter every point required without overlooking the first Pen Down (PD) move or the final Pen Up (PU) move on a new contour. The job will be either shown in the Digitize window or opened in JobPreviewer. Digitize files can also be imported into a CAD/CAM program or directly run back on the machine.

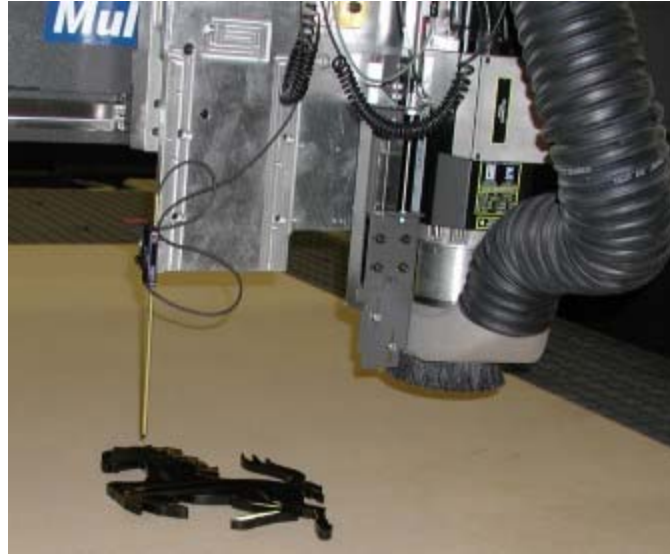
1. Make sure that JobNameServer  has been activated on the computer.
2. Press **Go Home**  or move to a location where 0,0 will be located and press **Set Home** .
3. Press **Menu**  to access the menu system.
4. Press **Jog Arrows**  until Digitize is highlighted in blue.
5. Press **Enter**  to start digitizing.
6. Enter a numerical name for the file to be created. When executing or importing the file, the operator must include a number before "dig.cnc" (e.g., 1234dig.cnc).
7. Jog to the start of a contour and press **Enter** . This carries the PU move to just above the first location. Digitize starts in a PU mode, so the first move is a PU move.
8. Press **Set Max Depth**  to toggle between PU and PD. This enters a PD move at the same X,Y.
9. Stay in PD mode.
10. Jog around the contour and press **Enter**  to add new points.
11. Press **3**  near the end of the contour. This will position the bit directly over the first point in the contour.
12. Press **Enter**  to enter the point.
13. Press **Set Max Depth**  to toggle to PU.
14. Jog to the next contour and repeat the process until all contours have been digitized.
15. Press **Go Home**  to go to 0,0 at the end of the job.
16. Press **Exit**  to end the job.

[Options](#)

Innovation. Quality. Performance. Best in Industry. Worldwide.

Rapid Shape™

The **MultiCam** Rapid Shape can automatically digitize both simple and complex shapes. Mounted on any **MultiCam** Router, this device enables the router to automatically follow an edge of a pattern or object placed on the router bed and record the outline on the host PC.



Easy and accurate calibration makes digitizing a variety of materials possible with digitizing speeds typically of 60 inches/minute. The Rapid Shape software generates a G-Code file for importing to many CAD/CAM systems.

[Options](#)

Using Rapid Shape™

Once JobNameServer has been verified on the system, Rapid Shape can be used in conjunction with the cutting sequence.

1. Make sure JobNameServer  is running on the computer.
 2. Press **Go Home**  or move to a location where 0,0 will be located and press **Set Home** .
 3. Press **Menu**  to access the menu system.
 4. Press **Jog Arrows**  until Digitize is highlighted in blue.
 5. Press **Enter**  to access the Digitize sub-menu.
 6. Set the 5 options for Rapid Shape: Scan Rate, Step Size, Scan Mode, Add Corners, and Output Start.
 - a. **Scan rate** sets the speed the system will auto scan. The default setting is 2.0" per second.
 - b. **Step Size** sets the distance along a line between 2 scanned points. The default setting is 0.15".
 - c. **Scan Mode** sets the style of scanning to be completed. Operators should select EDGE for templates and LINE for drawings or patterns.
 - d. **Add Corners** adjusts for corners when detected by the system.
 - e. **Output START** places "START" at the beginning of the G-Code file, which is required by some software packages due to the importance of the G-Code file.
 7. Move the cursor to "Start Digitize" and press **Enter** .
 8. Enter a numeric file name and press **Enter** .
 9. Press **Jog Arrows**  to place the **center of the bit** over the front right corner of the material.
 10. Press **Set Home** .
- 
11. Press **Enter**  to set the home position. This will store the absolute X and Y coordinate locations in the controller's memory.



12. Press **Jog Arrows**  to move the spindle to the edge of the pattern or template. This sets the direction Rapid Shape will go around the contour of the pattern or template.
13. Press **Five**  to start the Auto Scan. The system will move back and forth for a few seconds to calibrate and then start moving in the set direction along the contour of the pattern. The scanner will continue along the contour until the start point is reached, unless the operator interrupts the sequence.
- a. Press **Pause**  to stop the scanning process if needed.
- b. Press **Enter**  to access manual mode. The operator may continue Auto Scan by moving the spindle forward along the contour at least the step-size distance, while keeping the laser dot close to but not actually on the material, and pressing Auto Scan. The system will move back and forth to calibrate again before continuing around the contour. At the end of the Auto Scan, Digitize will be left in a PU move.
14. Repeat for any additional contours. The system will enter as many contours as needed.
15. Press **Cancel**  to exit out of Digitize mode once the part is completely scanned. To display the part on the computer screen, the operator can open the file in JobPreviewer. The scanned file will be displayed in the preview window.

Once the scanned file has been completed, it can then be either imported into a CAD/CAM package or executed at the machine through DNC, though the system will **not** complete tool offsets if the operator executes the scanned file from the machine.

[Options](#)

Rapid Shape™ Tips and Troubleshooting

1. **Scan rate** sets the speed the system will auto scan. The default setting is 2.0” per second. A slower scan rate will result in slightly better accuracy at a longer time, while a faster rate will take less time but have less accuracy.
2. **Step Size** sets the distance along a line between 2 scanned points. The default setting is 0.15”. The step-size distance should be small for detailed parts but can be larger for parts with less detail.
3. **Scan Mode** sets the style of scanning to be completed. Operators should select EDGE for templates and LINE for drawings or patterns.
4. **Add Corners** can be adjusted for corners when corners are detected during the cutting sequence.
5. **Output START** places “START” at the beginning of the G-Code file, which is required by some software packages due to the importance of the G-Code file.

Rapid Shape will exit if one of the following occurs:

*** The module is not loaded.**

The operator should load the module by selecting JobNameServer from the **MultiCam** folder under the Windows Start menu. Once the module is loaded, the router should be calibrated to the material to be scanned.

*** Rapid Shape found the start of the contour.**

The operator should move to the next contour to continue or cancel the sequence.



*** The operator pressed Pause**.

The operator can move farther along the contour and restart.

*** The system could not find the edge.**

The operator can move farther along the contour and restart.

*** The system lost a good edge.**

The operator can move farther along the contour and restart.

*** Rapid Shape has too many points.**

The operator can decrease the step size and start over.

*** The previous point is too close to Auto Scan.**

The operator can move farther along the contour and restart.

[Options](#)

Laser Pointer

The Laser Pointer option allows operators to set Home using the laser diode rather than relying solely on naked eye approximations. With the laser diode mounted on the carriage of the machine and pointed at the material surface, operators can see exactly where the cutting sequence will begin and set their Home locations exactly. Once the offset is established, operators can begin using the laser pointer to set Home. The offset will not need to be reset unless the laser diode is replaced or loses its orientation to the cutting head.

1. Turn on the Laser Pointer by pressing **0** . Operators can toggle the Laser Pointer ON and OFF* by pressing **0**, and the Home screen will show whether or not the Laser Pointer is active by inserting the letter **L** in the lower right corner. (Operators who wish to set Home with the standard method should leave the Laser Pointer OFF.) The following examples show how the X and Y values change when the Laser Pointer is toggled ON or OFF.

Laser Pointer ON



X= -0.000 Y= 0.000
Z= -0.500 Jog= MED
Feed%=100 RPM=18000
Tool= 1 Home= 0 L

Laser Pointer OFF

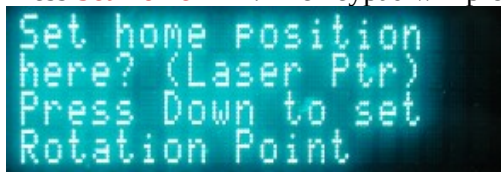


X= -2.549 Y= 4.921
Z= -0.500 Jog= MED
Feed%=100 RPM=18000
Tool= 1 Home= 0

* Operators cannot set Home using the Laser Pointer if the Laser Pointer is OFF.

2. Jog the Laser Pointer to the Home location.

3. Press **Set Home** . The keypad will prompt the operator to set the Home location using the Laser Pointer.



Set home position
here? (Laser Ptr)
Press Down to set
Rotation Point

4. Press **Enter**  to set Home.

5. Press **Enter**  to leave the Laser Pointer ON or **Cancel**  to turn the Laser Pointer OFF. If the Laser Pointer is left on, the laser beam will **not** follow the cutting head as it cuts since the Laser Pointer is offset from the center of the cutting head.



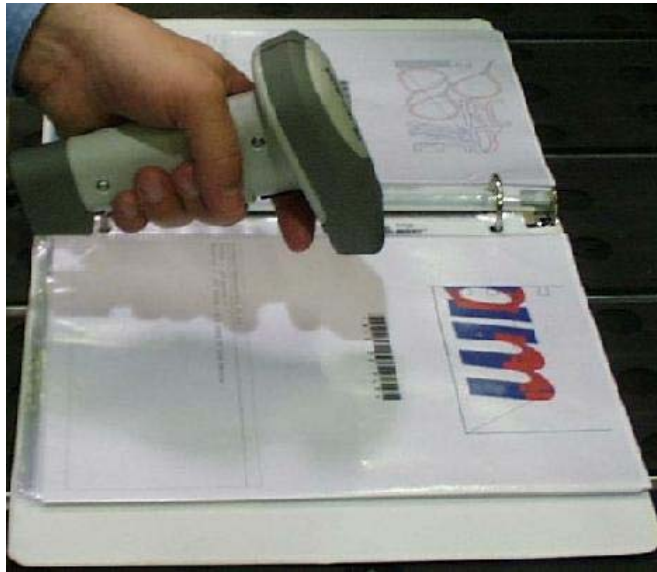
Turn off Laser Ptr?
ENTER=Yes, EXIT=No

6. Finish system preparations and run the file.

[Options](#)

Bar Code Scanner

The **MultiCam** Bar Code Scanning Interface has been developed to reduce setup time, minimize errors, and work directly with shop floor work orders. The bar code scanner works in direct conjunction with the DNC. Once the machine operator scans the bar code, DNC will automatically call up the correct file and send it to the controller. This saves time and reduces the chance for error.



An excellent tool for communication of special instructions, the work order contains a graphical representation of the cut file, tool specifications, material descriptions, and user notes. The bar code label on the work order ensures that the correct job is accessed.

[Options](#)

Chip Removal Options

Chip removal with any routing system is vital to machine performance, tool life, and the improvement of edge quality. **MultiCam** has developed different options to achieve chip removal needs in the routing industry. These options include the Pressure Foot and the Dust Collector, both of which remove unwanted waste material during and after a project. [Options](#)

Pressure Foot

The Pressure Foot can be used for both removing waste during the cutting operation and applying pressure to the surface of the material to reduce vibration and keep parts from moving. Its innovative design applies adjustable spring pressure, which makes the Pressure Foot an ideal option for cutting paper-masked plastics and acrylic materials. Equipped with a standard 2" hose input, the Pressure Foot is typically used with a high pressure vacuum and can lift with the spindle to avoid hitting clamps or other obstructions.

1. Use glide pad attachments when the material needs to be held down. Glide pad attachments are not recommended for materials with a fine finish. Brush attachments are a better choice for materials that scratch easily.
2. Adjust the height of the Pressure Foot so that the bottom of the glide pad is approximately 1/8" below the bottom of the routing bit when the spindle lifts.
3. Turn the wing nut on the height adjustment shaft to move the pressure foot up and down.
4. Re-tighten the wing nut once it is at the correct height.
5. Adjust the springs so that they are compressed halfway when the bit is at full depth in the material.



[Options](#)

Dust Collection Hoods

MultiCam can also provide a high flow dust collection hood for most of the collet and ATC spindles. The dust collector attachment is designed to be used with high CFM dust collection systems generally found in wood shops.

Dust Boot Dust Collector

The Dust Boot is designed to be used with a collet spindle in high CFM dust collection environments. With a 4" hose input, the dust boot moves with the Z-axis.



Pull-up Dust Collector for ATC Spindle

The Pull-up Dust Collector is designed for use in a high CFM dust collection environment. With a 4" hose input, the pull-up dust collector lifts when the spindle is not running and moves with the Z-axis.



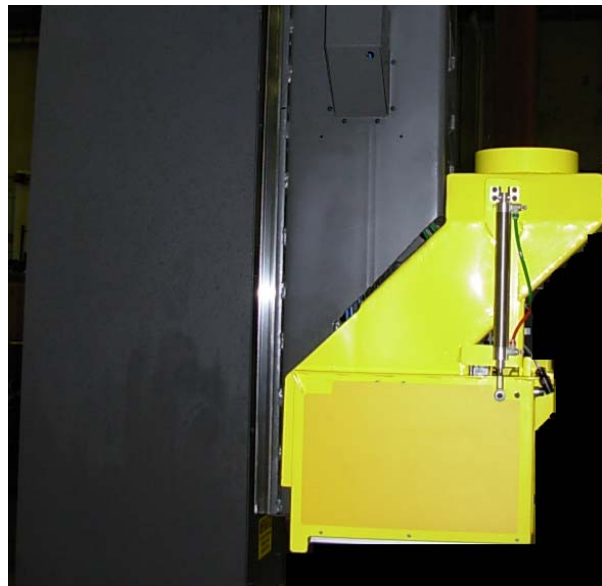
Dust Collector for ATC Spindle

The Dust Collector for the ATC Spindle is designed for use in a high CFM dust collection environment. With a 4" hose input, the dust collector lifts when the spindle is not running and moves with the Z-axis.



Pantleg Dust Collector

The Pantleg Dust Collector is designed to be used with machines that access cutting sequences on the C-axis with CFM dust collection systems. More specialized than other dust collectors, the Pantleg Dust Collector is equipped with an 8" input hose and the capability for maneuvering very large tools. The dust collector lifts up when the spindle stops moving, and all movement follows the Z-axis.



[Options](#)

Dust Collection Hoods Installation for the Dust Boot

Mounting brackets and clamps for the Dust Boot dust collection hood are located on both sides of the spindle. The operator must identify these prior to installing the dust collection hood.

1. Locate the pocket into which the spindle will fit on the top plate of the dust collection hood.
2. Place the dust collection hood under the spindle and lift it into the spindle.
3. Place the holding clamps over the mounting bracket and flip them into place.
4. Connect a 4" vacuum hose to the front of the hood. The standard dust collection hood has a 4" vacuum hose connection point, and adapters are available at any hardware store for proper connectivity if needed.
5. Provide enough hose for the dust collection hood to travel the entire length of the table. The hose should be suspended from above the table to prevent it from being pulled out or otherwise damaged as the system moves.

[Options](#)

Engraver

MultiCam offers an engraving spindle that is ideal for most engraving applications and only available with certain systems. The floating nose engraver is belt-driven directly from the router spindle, and the spindle accepts standard top-loading engraving cutters.

The engraving spindle is available with a standard 1/4" collet, though other collet sizes such as 11/64" are available.



[Options](#)

Pneumatic Drill

The Electric/Pneumatic Drill adds up to 2 additional drills to the **MultiCam** Router for additional production efficiency. The drill and spindle operate from the same job file, and no proprietary interface software is necessary to operate.

The Pneumatic Drill requires compressed air but operates at 3000 RPM on a standard 3/8" chuck.

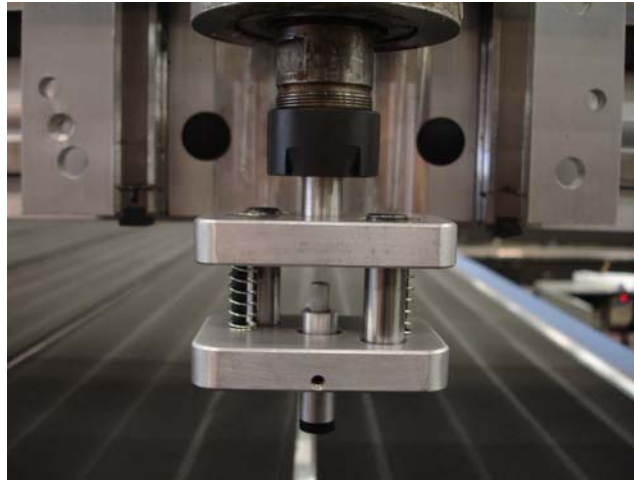


[Options](#)

Vinyl Knife

The Vinyl Knife option is available for all routers to cut thin materials, such as vinyl labels, following the same cut file as the router file. The vinyl knife can be inserted into the chuck just like a regular tool and can also be inserted into the tool changer for ATC machines. Operators can run regular cut files as well as test files using the vinyl knife, which is spring-loaded for more accurate pressure on the material.

When configuring the tool, operators will need to set the tool to No Spin to get a better quality performance. Operators must manually set surface with the vinyl knife and must always use extreme caution when handling the blade.



[Options](#)

Under Gantry Pusher

The Under Gantry Pusher is available on 5000 and 7000 Series machines and is used to remove material from the front of the machine. This option is most appropriate for operators who load the machines from the back of the machine and remove the material from the front.



[Options](#)

Heat Exchanger

The spindle heat exchanger cools the heated water from the spindle, which allows for repeatability on cut parts even in a production environment. When operators utilize a spindle heat exchanger, they can be sure of parts that are not too hot to handle and still in the proper dimension. The spindle heat exchanger should be selected based on the horsepower of the spindle, the ambient temperature of the production room, the type of coolant (e.g., water, glycol), and the spindle operating cycle.

Used with Radiator-Cooled Spindle	Used with Water-Cooled Spindle
	

[Options](#)

Cold Air Gun

The cold air gun focuses a blast of cold air at the bit while it cuts into the material. This method prevents any use of lubricant during production, as well as secondary cleaning once the material has been machined. Air-cooled bits allow the operator to move on to the next cutting sequence in less time, and the tool life of the bit is extended by almost 50%. The compressed air is low pressure and helps to clear away chips and dust.



[Options](#)

Table Format

The table option is available in different formats for all routers. The T-plate table is the preferred format for bolting down parts that have irregular shapes or parts that cannot be held in place securely with a vacuum. The vacuum T-plate is the preferred format for bolting down parts that react well to the vacuum but may be irregularly shaped. The gridded vacuum is the preferred format for holding down parts that have large, flat surfaces that respond well to a vacuum system.

T-plate Table
and
Vacuum T-plate Table



Gridded Vacuum Table



[Options](#)

Vacuum Options

The vacuum pump option is available with different levels of horsepower. The sales associate can help operators determine which type of vacuum pump and in what configuration would be the most effective for the machine and the material being cut.

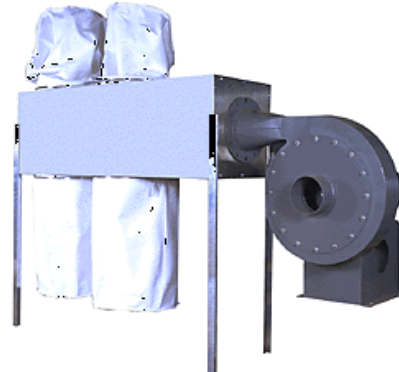
Positive Displacement (PD) vacuum pumps are available in 10hp, 20hp, and 30hp at 200 CFM, 425 CFM, and 700 CFM respectively. PD pumps are used mainly with nested and random part manufacturing.



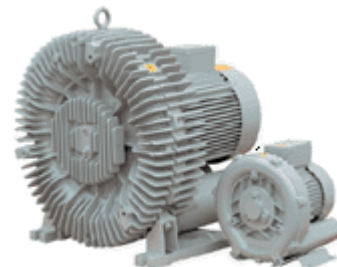
High Pressure vacuum pumps are available in 5hp, 10hp, and 15hp at 70, 177, 238 CFM respectively. While all three of the high pressure vacuum pumps are used mainly with fixture parts, the 15hp high pressure vacuum pump is also used with random part manufacturing.



High Flow vacuum pumps are available in 25hp at 3000 CFM. High flow vacuum pumps are used mainly with large tables that hold large parts.



Regenerative Blowers are available in 8.5hp, 17hp, and 33hp at 215 CFM, 353 CFM, and 776 CFM. Regenerative blowers are used mainly with random part manufacturing.

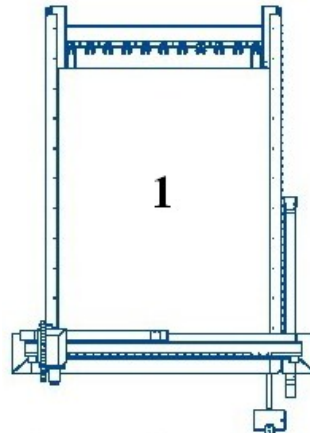


[Options](#)

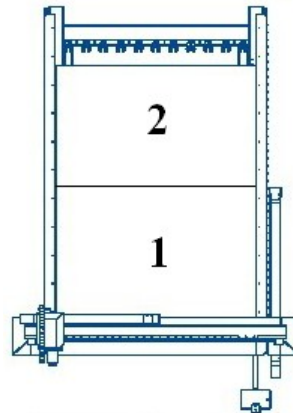
Vacuum Zones

The vacuum zone option is configured per table. Operators can select how the different parts of the table will receive the vacuum holding. The zones can be set up as 1, 2, 4, or custom.

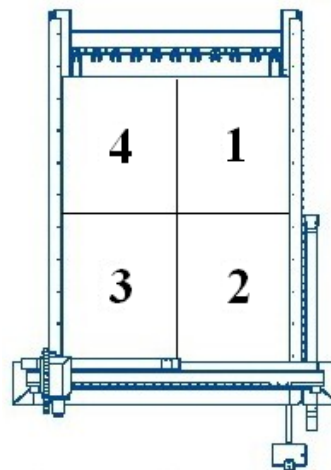
1 zone provides equal vacuum pressure to the entire table.



2 zone provides vacuum pressure to the top or bottom of the table.



4 zone provides vacuum pressure to the four corners of the table.

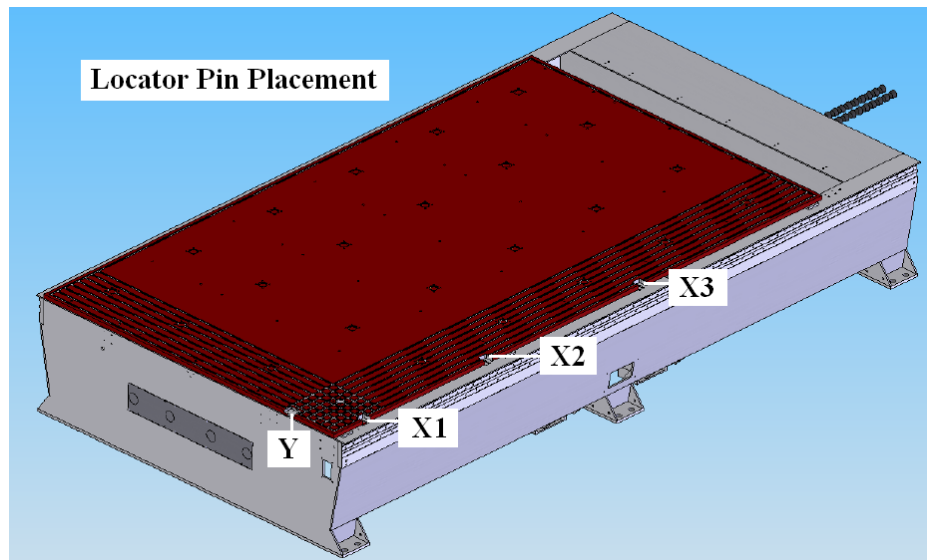


Custom zone can be modified per the operator's specifications.

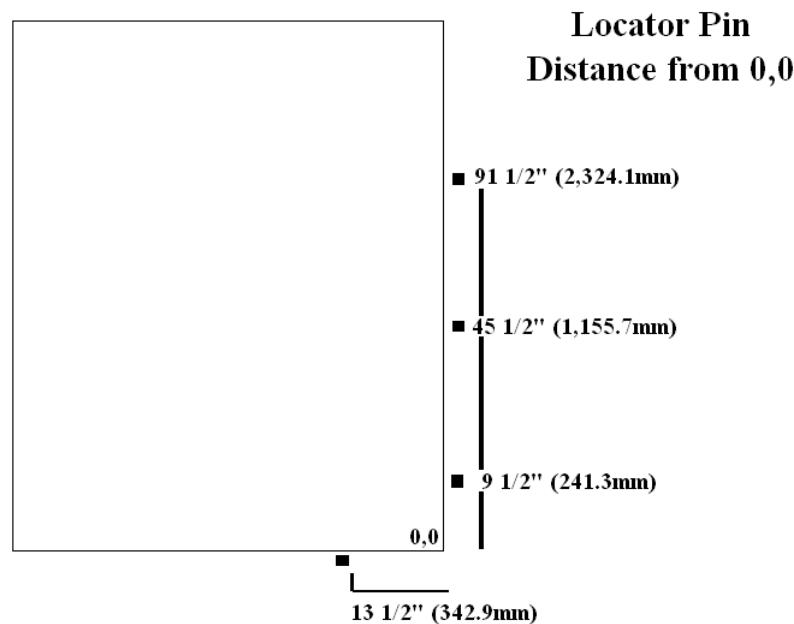
[Options](#)

Locator Pins

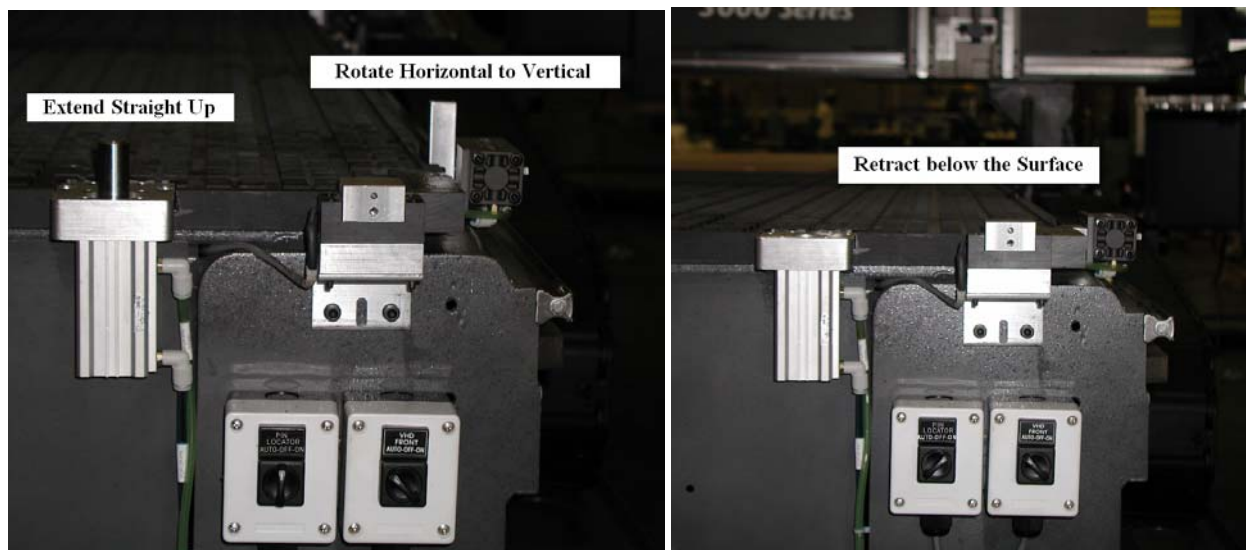
Operators have the option of requesting locator pins on their 3000 and 5000 Series Router machines. The locator pins allow the operator to maintain the material alignment at 0,0 when cutting features along the surface or through the material. Features can be guaranteed in the right place since the locator pins prevent any unintentional adjustments of the material during the cutting sequence.



Routers with the standard locator pin option will have one locator pin along the Y-axis and two or three locator pins along the X-axis depending on the size of the table and the customer preference. Standard locator pin placement is Y, X1, and X3 with the operator's ability to move the X3 locator pin to X2. Customers can choose different locator pin locations as needed. The standard distances are shown below.



Locator pins can either extend straight up and down 2" (51mm) or more above the table surface or rotate 2" (51mm) or more horizontal to vertical along the X-axis. The Y-axis locator pin as well as standard locator pins moves straight up and down, but wider tables may require locator pins that rotate horizontal to vertical.



[Options](#)

Safety Mat

The Safety Mat is a pressure-sensitive foot mat that will cause the machine to enter into Pause mode when the mat is stepped on during machine operations. This Pause mode can only be cleared by pressing the “Safety Mat Reset” switch at the operator station and then clearing the error at the keypad.

MultiCam advises operators to pause operations during any changing of tools or materials and recommends that operations be paused when any disruptive action (e.g., change of shift, emergency situation) occurs in the cutting area. All cutting operations should be stopped and the machine turned off before any maintenance issues are addressed.

[Options](#)

Light Curtains

The light curtain is a useful tool for operators as a secondary way to observe safety on the production floor. The setup and configuration of the light curtain vary for each machine and circumstance. The light curtains utilized by **MultiCam** are horizontal, and customers may use their own vertical light curtains in conjunction during production.

ALL SAFETY MEASURES ADOPTED BY THE CUSTOMER ARE AT THE CUSTOMER'S DISCRETION AND DONE SO AT THE CUSTOMER'S RISK.

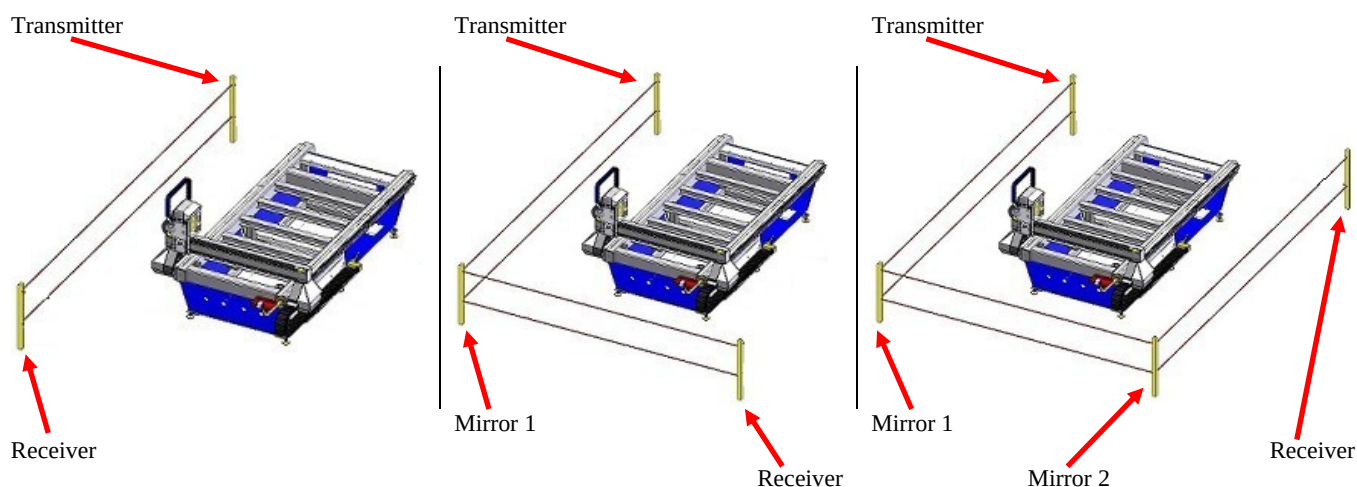
MULTICAM HAS NO RESPONSIBILITY REGARDING SAFETY FEATURES.

IF THE TECHNICIAN INSTALLS THE SAFETY DEVICE, IT IS DONE AS A COURTESY.

The following requirements must be met in order for the light curtain to be effective.

- Customers must guarantee some specific settings and configurations before a light curtain can be installed on an existing machine. These specifications are outlined in the light curtain user manual, which is provided in hardcopy with the machine and available in softcopy upon request.
- All applicable federal, state, and local government requirements must be followed.
- All operators and machine personnel must be knowledgeable about the machine's processes and safety risks.

One light curtain has a 70m (229.6') linear range. The inclusion of mirrors derates this distance by 12% per mirror (e.g., 1 mirror = 61.6m {202'} linear range, 2 mirrors = 47.7m {156.4'} linear range). With a maximum of 2 mirrors, the light curtain offered by **MultiCam** will address up to 3 sides of the machine. The available single light curtain configurations are shown below.



MultiCam advises operators to pause operations during any changing of tools or materials and recommends that operations be paused when any disruptive action (e.g., change of shift, emergency situation) occurs in the cutting area. All cutting operations should be stopped and the machine turned off before any maintenance issues are addressed.

[Options](#)

Coolubricator System

Coolubricator Systems are completely self-contained, positive-displacement, continuous spray systems that operate when the adjustable pulse generator automatically cycles the adjustable metering pump(s). Operators just have to supply an air signal to operate the system.



The liquid output can be adjusted with the brass knob located on each metering pump, and any amount of air can be used to atomize the liquid. A brass knurled air metering screw controls the flow of air atomizing out of the nozzle, which determines the density and distance of the spray. The Coolubricator System offers complete control of the spray as well as accurate placement of the lubricant or other material being applied. The spray output includes an approximately 15-20 degree angle depending on the amount of air introduced. Operators can adjust the area covered by using the air and liquid adjustments.

The Coolubricator System allows for multiple metering pumps to be stacked with each pump supplying its own nozzle. This configuration allows for greater spray coverage from multiple nozzles, which are ideal for multiple spindle drills and taps, stock punching, or any application requiring material being sprayed from several points. The nozzle outputs are controlled by independent adjustments of the air metering screw and metering pump stroke adjustment knob. Fluid output may be increased by using 3-drop output pumps in place of standard 1-drop pumps, though operators are encouraged to discuss this with the manufacturer beforehand.

[Options](#)

Operating the Coolubricator System

Operators of the Coolubricator System should be careful when installing and operating the auto misting system.

1. Determine the position and method of attachment – either directly or with optional magnets – before mounting the Coolubricator System to the application equipment.
2. Check for any obstructions that may pinch or kink liquid or air feed lines.
3. Mount the Coolubricator in close proximity to machine application either directly to the machine or with the optional magnets to the right and left flanges of the Coolubricator case or bottom of the case.
4. Mount and position the nozzles as desired.
5. Attach the air supply line to 1/4" FPT inlet on the air filter supplied with steel enclosure if enclosed and directly to the top manifold block if non-enclosed.
6. Make sure the reservoir or remote tank Low Level Switch is hardwired. This step is optional and can be skipped.
7. Wire the solenoid valve or connect the air pilot valve.
8. Fill the gravity feed reservoir or air trap.
9. Loosen the drain plug until liquid flows free of air.
10. Tighten the drain plug and cycle unit. On units supplied with fluid from a pressurized supply tank, set the supply tank pressure at 5-10 PSI and depress the air vent cap until the trap is 75% full.
11. Turn the brass adjustment knob to the maximum output setting and turn the air supply on and off with the optional manual ON/OFF valve or by manually depressing override cap. Operators may either start the machine tool with a remote switch, which was installed to energize optional electric solenoid, or activate the air signal to the air pilot valve.
12. Adjust the Pulse Generator to 6-8 pulses per minute.
13. Close the air metering screw when fluid is seen coming out of nozzle supply tube with each cycle. Open the air metering screw to the desired degree of atomization. Operators should use a minimum amount of air atomization to spread the lubricant and deliver it to the application area.
14. Use the Pulse Generator as the Primary Fluid Adjustment.

[Options](#)

Troubleshooting the Coolubricator System

Most operating problems with the Coolubricator System can be easily corrected. The operator should contact the manufacturer should any problem occur that is not addressed in this document.

Symptom	Cause	Remedy
No fluid output following reservoir evacuation	Pulse generator is set too slow or too fast.	1. Check the reservoir level. 2. Use full air line inlet pressure and check for stroking of the meter. Set to max. 3. Check for fluid movement in the capillary tubing at each meter stroke. 4. Loosen the 5/16" brass hex bleed screw 2 revolutions until only fluid comes out. Close the screw tightly and recheck the meter output. 5. Check inlet air and set to max.
Reduced fluid output	Non-lubricating fluids like water and solvents can cause premature wear of internal parts.	Check the air inlet pressure. Set to max so that meters operate crisply.
Pulse generator not functioning	Obstruction in the Coolubricator System	Remove the pulse generator and check the air inlet screen for particular matter.
Slow or sluggish meter operation	Broken spring on an inadequately lubricated U-cup	1. Make sure the air inlet pressure is set at max. 2. Rebuilt the meter.
Meter depressed and not returning	Piece of debris in the fluid causing the liquid pin to jam	1. Disassemble and reassemble the auto mister and look for broken parts. 2. Rebuild the meter if a return spring is broken.
Fluid flowing continuously out of the nozzle or Reservoir emptying without the system running	Check valve seal not properly functioning	Check the seat and seal spring for cleanliness.
Fluid accumulation in the outer tubing	Insufficient atomizing air exiting the nozzle	Increase the atomizing air flow or direct the nozzle coupler outlet downward to remove excess fluid.
Air bubbling upward into fluid reservoir	Possible leakage in the valve seat	1. Check the valve seat and seal for any signs of leakage. 2. Use a rebuild kit and replace the static sealing O-ring if necessary

[Options](#)

Innovation. Quality. Performance. Best in Industry. Worldwide.

Connections

[top](#)

Communication	2
Ethernet	
Ethernet Connection Information	2
Setup for Ethernet Connection	8
Ethernet Cables	10
Serial	
Setup for Serial Connection	11
Communication Troubleshooting	14
Communications Port Settings of Windows	16
Serial Line Amplifier	18
Serial Cable Construction	19
Serial Communication	19

Communication

The **MultiCam** Controller will support both Ethernet and Serial communications; however, **MultiCam** recommends using Ethernet communications whenever possible. [Connect](#)

Ethernet Connection Information

TCP/IP Ethernet connection is the preferred form of communications to all **MultiCam** controllers. The following information discusses several key components of Ethernet connections and how those components relate to connecting personal computers (PCs) and **MultiCam** CNC machines.

Requirements

Any company choosing to utilize **MultiCam** Ethernet communications as recommended must meet the following system requirements:

- Network card installed in the computer
- TCP/IP software installed in the computer
- Ethernet 10/100 MHz multiple port switch (cannot be wireless or a router)
- Ethernet patch cabling
- **MultiCam** Controller IP/Subnet in the same class network (IP/Subnet) as the Host PC

Wireless communication is not recommended for any **MultiCam** Ethernet system.

Ethernet Process

Every computer on a network has an IP address. The IP address tells other computers that the Host PC is ready to communicate with another device on the network. Each computer IP address must be unique within that network (e.g., 192.168.0.176 is the IP address for only one computer on the network).

In addition to IP addresses, each computer on a network is also assigned a subnet address. The subnet address is simply a way of separating larger networks into smaller, more manageable ones (e.g., 255.255.255.0 is one subnet address for **MultiCam** operating systems). Additionally, **MultiCam** CNC machines by design can only be used on a class "C" Network.

Setup

When any technician visits a customer site, he or she must first determine what type of network is available for the connection. The 2 types of networks are described below.

- Dynamic Host Configuration Protocol (DHCP) Network – Automatic Configuration

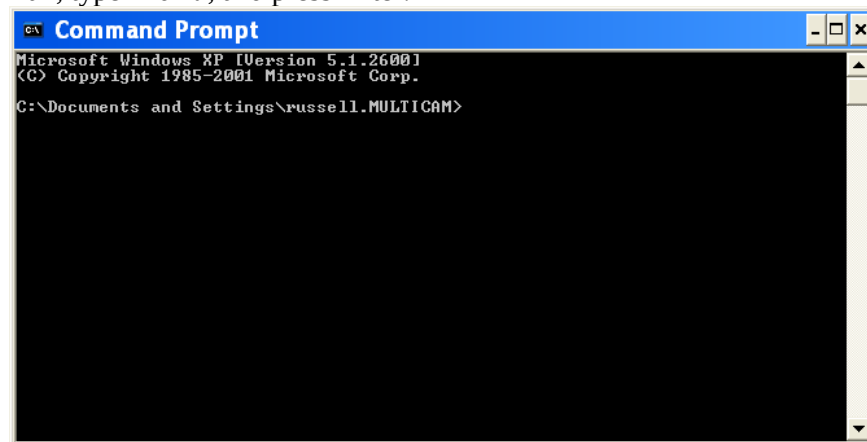
The DHCP server will automatically provide networked computers with the required TCP/IP configuration from a central location. When a computer requires the use of TCP/IP network resources, it broadcasts a request for address information. The DHCP server responds to this request by assigning a new address and sending it to the computer along with other required configuration information for the network. This information is acknowledged by the computer and used in setting up that configuration.

- Static Network – Manual Configuration

The static network has a manually configured setup for each computer or device, and each device must be assigned the same subnet mask but a different IP address from all other computers and devices.

By default, most laptops and computers are shipped with DHCP already enabled. This does NOT mean that the network has a DHCP server that will automatically assign the computer an individual IP address. Technicians should still complete the following to determine the proper configuration for the machine:

1. Use the command prompt in Windows to check the IP configuration. Technicians can use either of the following methods:
 - a. Go to Start >> All Programs >> Accessories >> Command Prompt.
 - b. Go to Start >> Run, type in *cmd*, and press Enter.



2. Type in `ipconfig /all` with a space between the `g` and the forward slash. This will pull up the Windows IP configuration for the computer.

```

C:\WINDOWS\system32\cmd.exe
Microsoft Windows XP [Version 5.1.2600]
(C) Copyright 1985-2001 Microsoft Corp.

C:\Documents and Settings\bmars\MULTICAM.000>ipconfig /all

Windows IP Configuration

    Host Name . . . . . : ENG13
    Primary Dns Suffix . . . . . : multicom.local
    Node Type . . . . . : Hybrid
    IP Routing Enabled. . . . . : No
    WINS Proxy Enabled. . . . . : No
    DNS Suffix Search List. . . . . : multicom.local
                                     multicom.local

Ethernet adapter Local Area Connection:

    Connection-specific DNS Suffix . : multicom.local
    Description . . . . . : Intel(R) PRO/100 UE Network Connecti
on
    Physical Address. . . . . : 00-11-25-6C-19-95
    Dhcp Enabled. . . . . : Yes
    Autoconfiguration Enabled . . . . . : Yes
    IP Address. . . . . : 192.168.1.64
    Subnet Mask . . . . . : 255.255.254.0
    Default Gateway . . . . . : 192.168.0.1
    DHCP Server . . . . . : 192.168.0.7
    DNS Servers . . . . . : 192.168.0.7
                           192.168.0.18
    Lease Obtained. . . . . : Tuesday, August 05, 2008 9:46:29 AM
    Lease Expires . . . . . : Wednesday, August 13, 2008 9:46:29 A
M

C:\Documents and Settings\bmars\MULTICAM.000>

```

3. Look for the IP address and subnet mask in the `ipconfig /all` screen (i.e., DHCP Enabled status and the DHCP Server address). This information will be required if the technician has to change any of the addresses.

```

C:\Command Prompt
Microsoft Windows XP [Version 5.1.2600]
(C) Copyright 1985-2001 Microsoft Corp.

C:\Documents and Settings\russell\MULTICAM>ipconfig /all

Windows IP Configuration

    Host Name . . . . . : TECH12
    Primary Dns Suffix . . . . . : multicom.local
    Node Type . . . . . : Mixed
    IP Routing Enabled. . . . . : No
    WINS Proxy Enabled. . . . . : No
    DNS Suffix Search List. . . . . : multicom.local
                                     multicom.local

Ethernet adapter Local Area Connection:

    Connection-specific DNS Suffix . : multicom.local
    Description . . . . . : Broadcom NetXtreme Gigabit Ethernet
    Physical Address. . . . . : 00-80-E4-C3-6A-A7
    Dhcp Enabled. . . . . : Yes
    Autoconfiguration Enabled . . . . . : Yes
    IP Address. . . . . : 192.168.1.74
    Subnet Mask . . . . . : 255.255.254.0
    Default Gateway . . . . . : 192.168.0.1
    DHCP Server . . . . . : 192.168.0.7
    DNS Servers . . . . . : 192.168.0.7
                           192.168.0.18
    Lease Obtained. . . . . : Wednesday, August 06, 2008 3:09:54 P
M
    Lease Expires . . . . . : Thursday, August 14, 2008 3:09:54 PM

Ethernet adapter Local Area Connection 2:

    Media State . . . . . : Media disconnected
    Description . . . . . : Bluetooth LAN Access Server Driver
    Physical Address. . . . . : 00-14-A4-DD-01-7C

C:\Documents and Settings\russell\MULTICAM>_

```

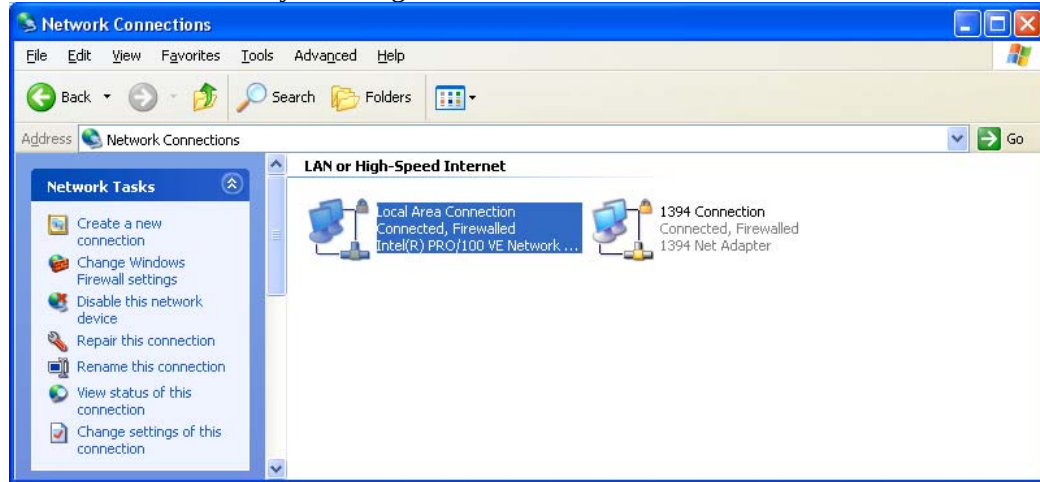
- If the network has an active DHCP, then DHCP Enabled will show Yes. The technician must also have an assigned IP address for the DHCP server (shown above).
- If the network does not have an active DHCP, then DHCP Enabled will show No, which indicates a static network.
- If any of the above information is not provided, then the technician is working with a static network.

Static Network Changes

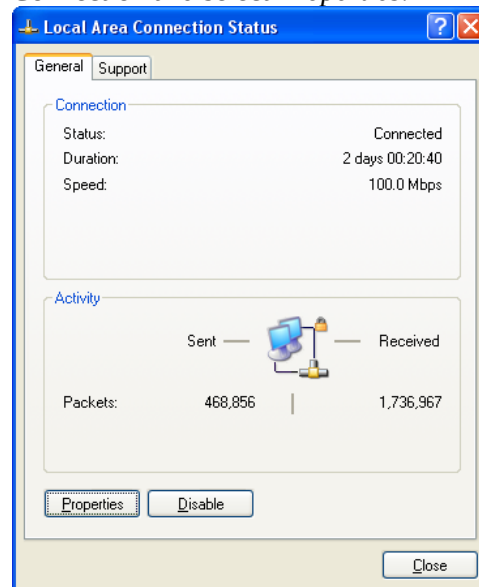
Technicians working with a static network may have to change the static IP and/or subnet address of the computer, the controller, or both. The following directions describe how technicians can change the IP and subnet addresses:

Computer Static IP/Subnet Address

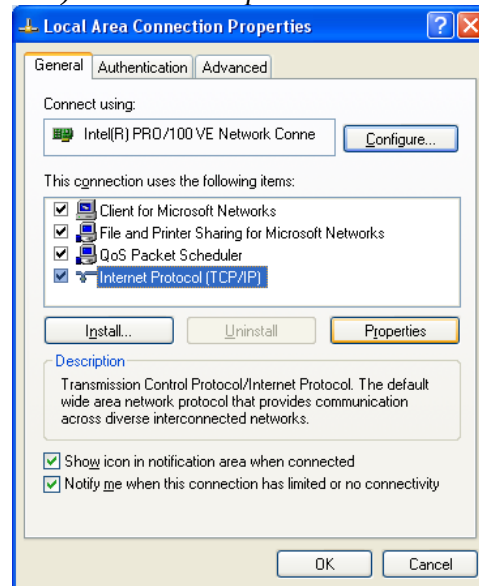
1. Access Network Connections by selecting Start >> Control Panel >> Network Connections.



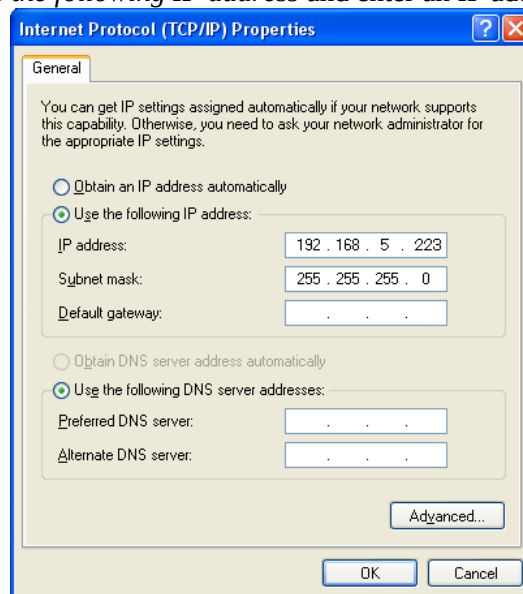
2. Double-click on the Local Area Connection and select *Properties*.



3. Highlight *Internet Protocol (TCP/IP)* and select *Properties*.



4. Click in the circle next to *Use the following IP address* and enter an IP address and the subnet mask.



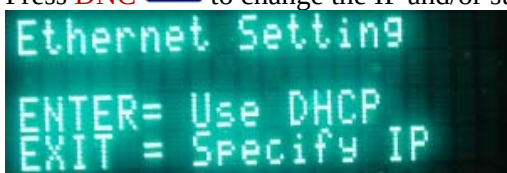
Controller Static IP/Subnet Address

1. Press **Shift** , **Help** , and **Two**  to access the Ethernet Information screen on the keypad.



```
Ethernet Information
IP :192.168.5.223
Mask:255.255.255.0
Host:192.168.5.101
```

2. Press **DNC**  to change the IP and/or subnet mask address.



```
Ethernet Setting
ENTER= Use DHCP
EXIT = Specify IP
```

3. Press **Exit**  to specify the IP address.

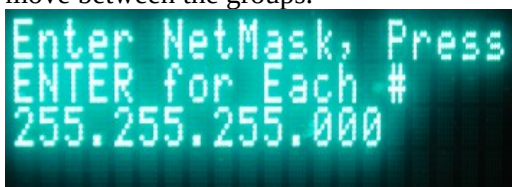
4. Type in the IP address as 4 groups of 3 numbers (e.g., 192.168.005.223) and press **Enter**  to move between the groups.



```
Enter IP, Press
ENTER for Each #
192.168.005.223
```

5. Press **Enter**  and wait for the screen to prompt for the netmask.

6. Type in the netmask address as 4 groups of 3 numbers (e.g., 255.255.255.000) and press **Enter**  to move between the groups.



```
Enter NetMask, Press
ENTER for Each #
255.255.255.000
```

7. Press **Enter**  to save the changes.

8. Reboot the machine for the IP and/or subnet mask address changes to take effect.

[Connect](#)

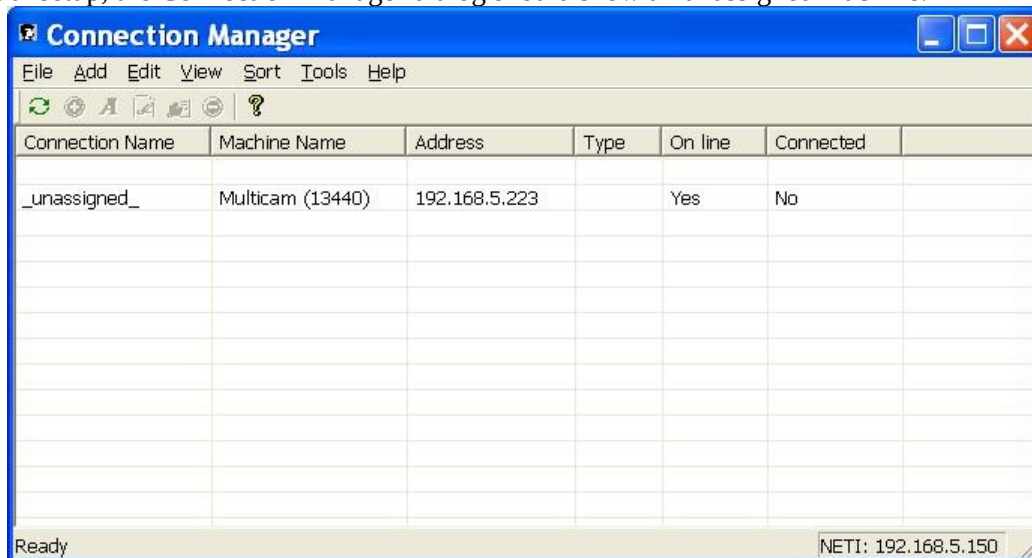
Setup for Ethernet Connection

The Ethernet communication is set up through the Machine Tool Properties applet. Operators should make sure that the CNC machine is on and connected to the computer via an Ethernet connection before attempting to automatically install the Ethernet connection. The following directions assume that the operator has complied with all hardware requirements for Ethernet connections (see **Ethernet Connection Information**).

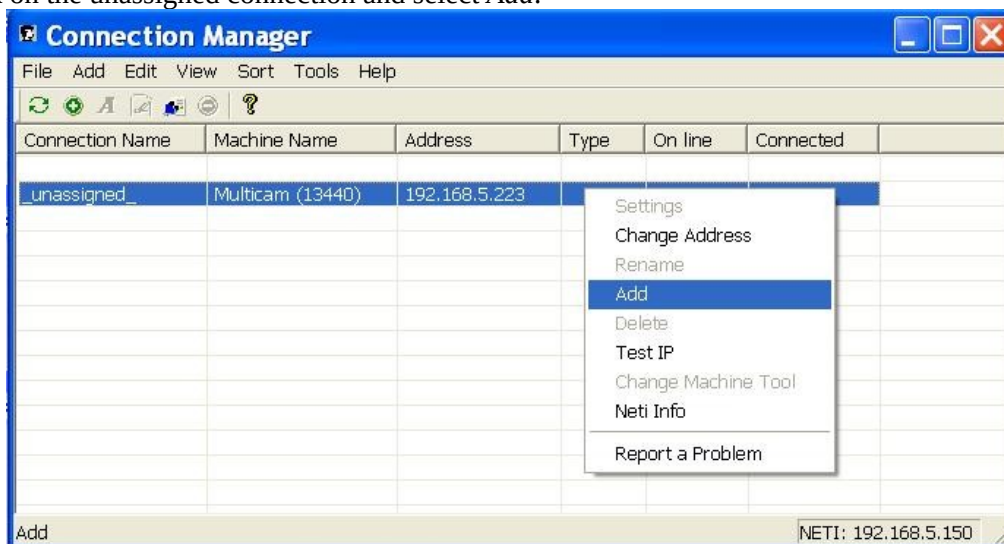
1. Right-click on the Machine Connections icon in the toolbar and select Connection Manager.



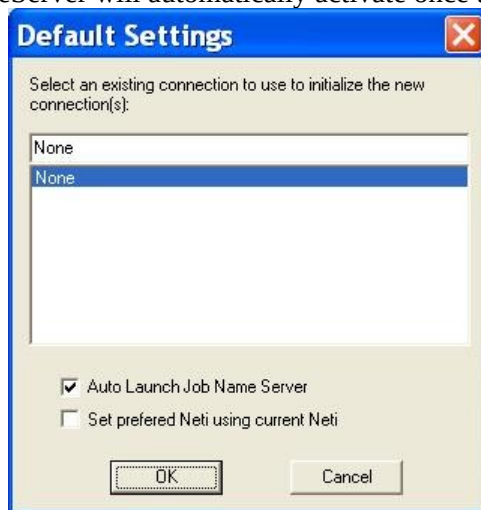
At the initial setup, the Connection Manager dialog should show an unassigned machine.



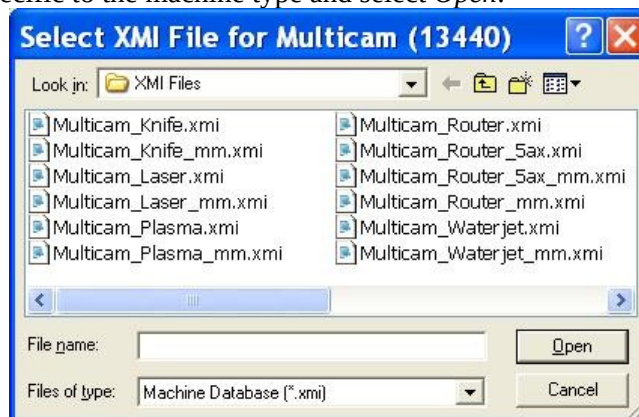
2. Right-click on the unassigned connection and select *Add*.



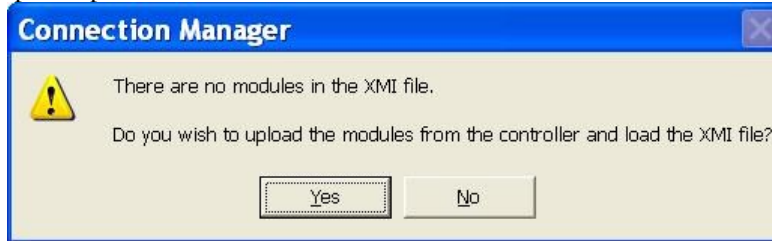
3. Select *None* at the Default Settings dialog. All customers are encouraged to check the box next to *Auto Launch Job Name Server* so that JobNameServer will automatically activate once the machine is connected.



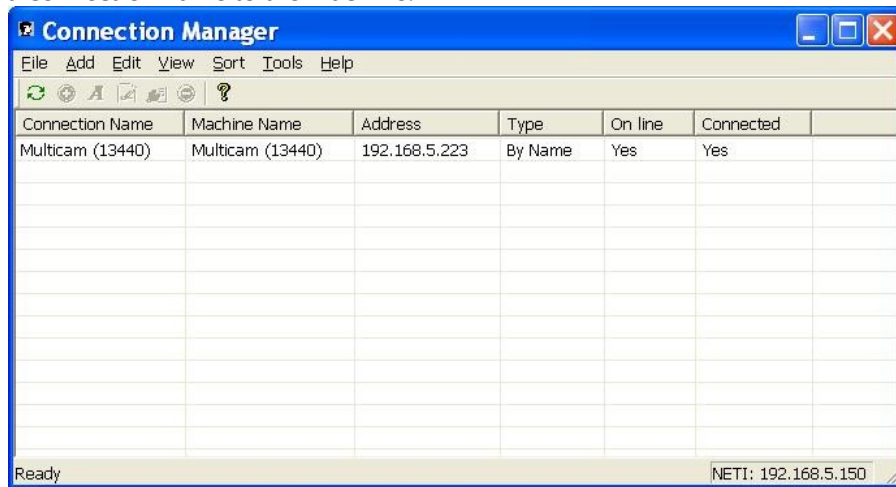
4. Highlight the XMI files specific to the machine type and select *Open*.



5. Select **Yes** at the prompt to upload the modules from the controller and load the XMI file.



This will assign a connection name to the machine.



[Connect](#)

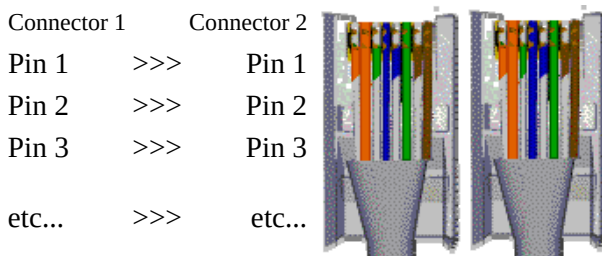
Ethernet Cables

If the company has an established network, then the computers are probably already connected through a hub or switch. A patch cable is a standard cable used to connect a network device consisting of a computer, a network printer, and a **MultiCam** Controller to a hub or switch. However, the patch cable is unable to provide a direct connection with the **MultiCam** Controller, so the host PC must be connected through a switch or hub.

RJ45 Patch Cable

A patch cable is also known as a straight-through cable. The wires are in the same order on connector 1 as they are on connector 2. The RJ45 connectors are used on each end of the cable, and operators must verify that the connectors are being held the same way when checking.

[Connect](#)



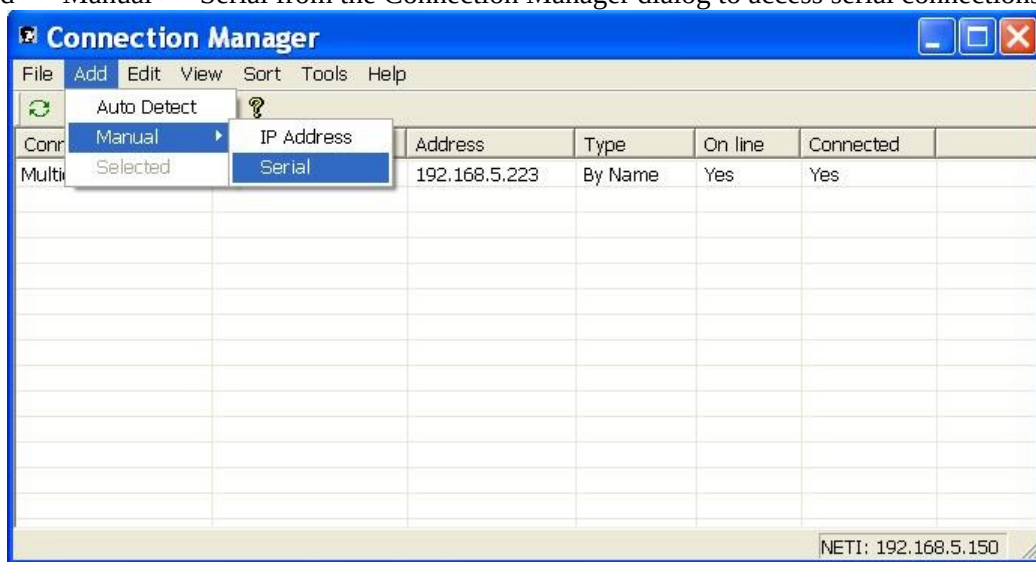
Setup for Serial Connection

The serial communication is set up through Machine Tools. Ethernet communication is always recommended over serial, but operators may need to use serial connections for troubleshooting or quick-fixes for communication.

1. Right-click on the Machine Connections icon in the toolbar and select Connection Manager.

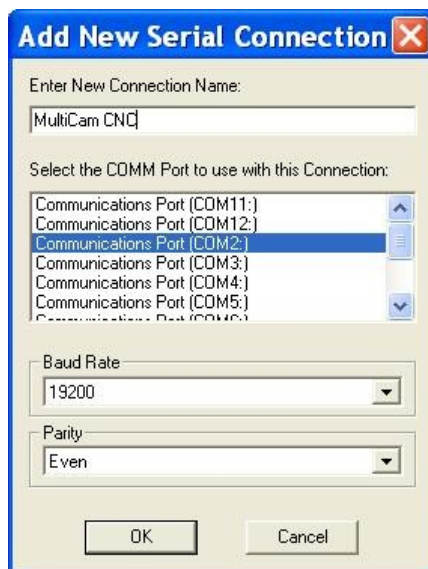


2. Select Add >> Manual >> Serial from the Connection Manager dialog to access serial connections.

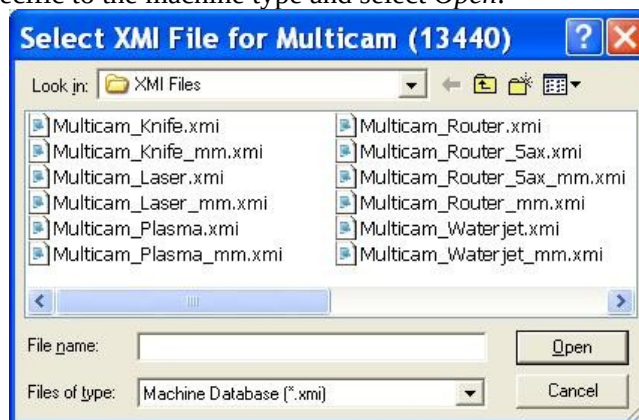


3. Enter or select the following information and select **OK**:

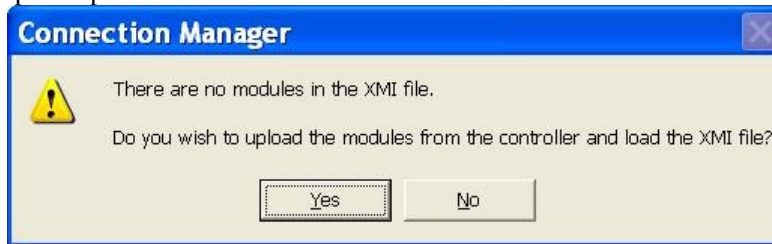
- Connection Name (e.g., MultiCam CNC)
- Communications port (e.g., COM2)
- Baud Rate – 19200
- Parity – Even



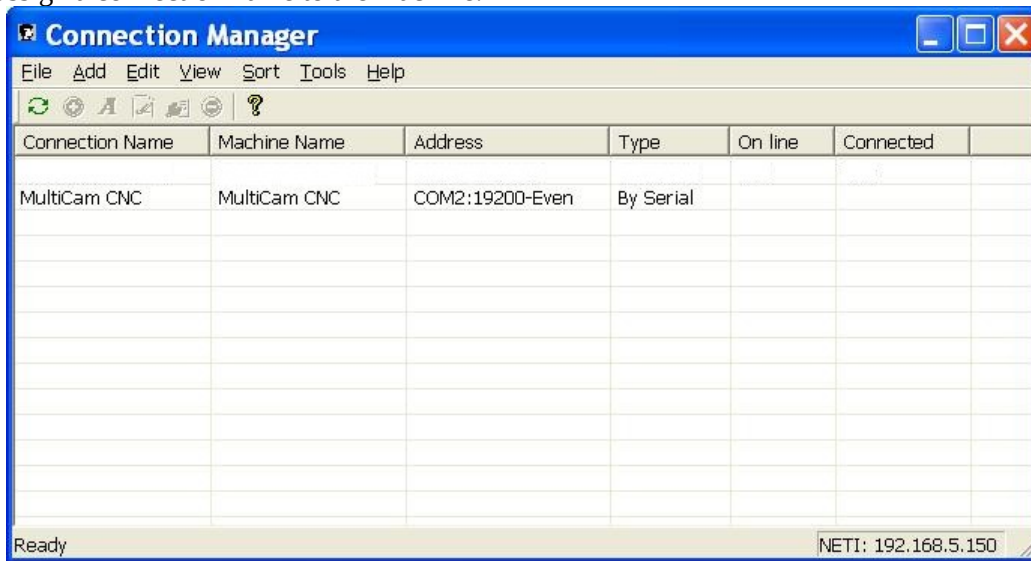
4. Highlight the XMI files specific to the machine type and select *Open*.



5. Select Yes at the prompt to upload the modules from the controller and load the XMI file.



This will assign a connection name to the machine.



[Connect](#)

Communication Troubleshooting

The **MultiCam** CNC machine is shipped with no connectors, and operators should purchase a 25' serial cable containing 9 wires to connect the **MultiCam** Controller to the Host PC. Data is sent from the computer to the machine controller. The data leaves the computer, passes through the computer Com Port or serial port, and moves through the serial cable and into the controller. Occasionally, operators experience difficulty when the machine does not accept a file, is unable to access DNC, loses connection, never establishes connection, or cuts off in a straight line in a manner that is not part of the original output file.

Most Common Reasons for Communication Failure

If the machine works some of the time but experiences random problems, then the machine may be experiencing some sort of communication problem. Here is a list of the most common problems for communication failure. Operators are encouraged to refer to this file before calling the local support technician or the **MultiCam** Technical Support.

- * The Windows Com Port settings, especially the FIFO buffer, are not correct. See Communications Port Settings of Windows.
- * The serial cable is longer than 100ft and is used without a Line Amplifier. See Serial Line Amplifier.
- * The serial cable has been operated next to high voltage or fluorescent lights.
- * The cable has been nicked, cut, or damaged in some fashion. See Serial Cable Construction.
- * The wrong Com Port has been selected on the host PC.
- * The Com Port IRQ conflicts with a modem installed in the host PC.
- * The internal ribbon cable inside the machine control cabinet has been damaged or has lost its connection.
- * The Max202 chip on the control board has failed.

Troubleshooting

The process of elimination is often used when troubleshooting various communication problems as more than one problem may have caused the current machine difficulty and different types of machines will respond differently to similar communication problems. The serial ports in the machine and in the computer could be damaged by a bad electrical storm or a comparable natural occurrence, and this damage will not always be fixed the same way for each machine. Below are common sites for communication problems and suggested ways to repair the damage.

Port Settings should be checked to verify that port settings and communication settings are correct.

* Machines that were manufactured after January 1996 have the following communication settings:

Baud rate: 19200; parity: even; data bits: 8; stop bits: 1

* Machines that were manufactured before January 1996 have the following communication settings:

Baud rate: 9600; parity: none; data bits: 8; stop bits: 1

Cable Testing 1 can be started with unplugging the serial cable from the side of the machine and plugging it back in before unplugging the serial cable from the back of the computer and plugging it back in. If the communications are still not working, Cable Testing 2 may help.

Cable Testing 2 involves opening the control box to inspect the cable connection extending from the side of the box where the external serial cable is plugged into the control board. This cable will be a ribbon cable plugged into the control board labeled G960, M10, M23 or M24. The connection going into the board can be unplugged and plugged back in. On many systems, there is an additional small ribbon cable extending from the top control board to the bottom control board that can also be tested. If the communications are still not working, Computer Port Testing 1 may help.

Computer Port Testing 1 for a computer with 2 serial ports involves plugging the machine into the other serial port and changing the port setting in DNC or the machine file output software. **Computer Port Testing 1** for a system with a serial port mouse involves plugging the mouse into the old port associated with the CNC machine. Windows should find the mouse automatically. If the machine works and the mouse does not, then the computer Com Port is probably bad and will need to be repaired by the computer vendor. If the communications are still not working, Computer Port Testing 2 may help.

Computer Port Testing 2 involves unplugging the serial cable from the side of the machine control box and plugging the loop-back plug provided in the tool box into the open end of the serial cable. After going to a DOS prompt, the operator should change directories to the mc directory (cd\mc) and run COMCHK.EXE. This will test the serial port and the cable. If the communication status is bad, errors will display on the screen. These errors could point to a faulty cable or a computer serial port. The test can be ended by pressing the ESC key. If the communication status is good, a continuous line of dots will appear along the bottom of the screen. If this happens, Machine Port Testing may help.

Machine Port Testing involves unplugging the serial cable from the computer and plugging the loop-back plug provided in the tool box into the open end of the serial cable. The collet and cover nut from the spindle should be removed, and the machine should be turned off and back on. If the machine enters self-test and begins to move, then the machine communications and serial cable are okay. At this point, the operator can probably assume there must be a problem with the computer Com Port or associated settings. If this process does not work, then 1 of 3 things have happened:

1. The serial port chip is bad and needs to be replaced. **MultiCam** can be contacted for the chip and installation instructions.
2. The internal ribbon cable connections have been damaged.
3. There is no self-test file loaded in the controller. **MultiCam** can be contacted for additional Help.

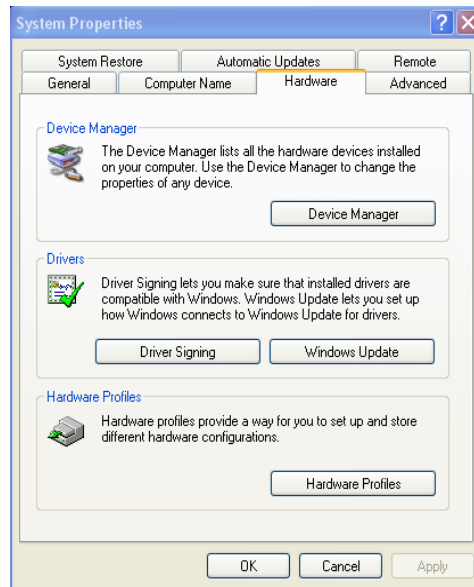
[Connect](#)

Communications Port Settings of Windows

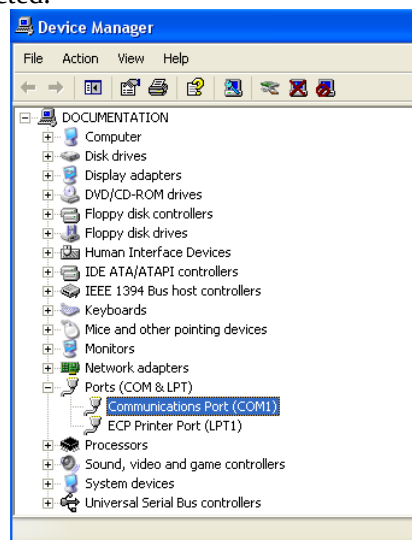
Most PCs are running Windows 2000 or higher as the Operating System. The default configuration of Windows is set to take advantage of larger (16 byte) FIFOs in the COM ports.

The **MultiCam** Controller has 4 to 7 character-receive FIFOs, which is less than half the standard default setting. Operators should reduce their Transmit FIFO levels when using a **MultiCam** Controller to lessen the chance for poor communications or loss of data that would result from having the settings too high.

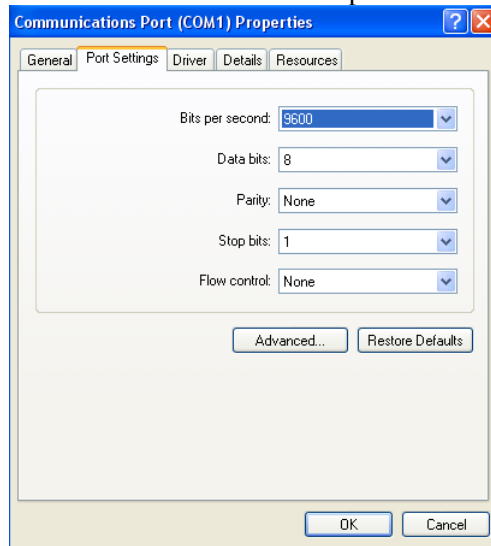
1. Right-click on the My Computer icon from the Windows Desktop and select the Hardware tab. Select *Device Manager*.



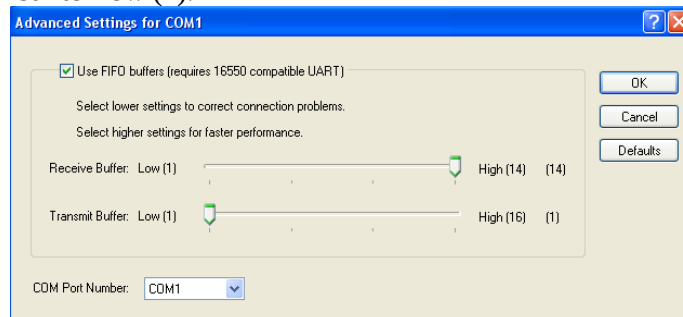
2. Select Ports (COM & LPT) and then select Communications Port (COM1) or the port to which the **MultiCam** Controller serial cable is connected.



3. Select the Port Settings tab from the Communications Port Properties window and select *Advanced*.



4. Drag the Transmit Buffer bar to Low (1).

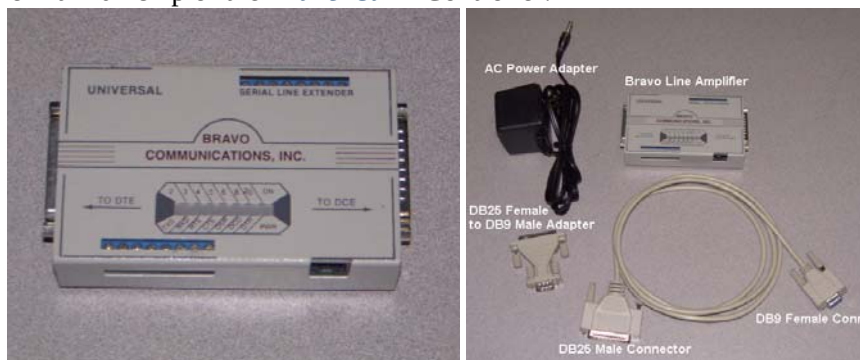


5. Select *OK* to all the windows that were opened and restart Windows to activate this setting.

[Connect](#)

Serial Line Amplifier

MultiCam recommends using the Bravo Line Amplifier or a comparable serial line amplifier and requires a line amplifier when the serial cable reaches or exceeds 100'. Not using the line amplifier will create communication problems between the host PC and the **MultiCam** CNC machine as too much load will be placed on the Com Port as well as on the Max202 chip of the **MultiCam** Controller.



The installation of the serial line amplifier involves the AC power adapter, line amplifier, DB25 female to DB9 male adapter, a 6' adapter cable with a DB25 male connector and DB25 female connector, and a serial cable with a DB9 male connector and DB9 female connector.

1. Flip the line amplifier over and make sure that the 2 switches are positioned towards the middle of the line amplifier.



2. Place the DB25 male connector of the 6' adapter cable into the "To DTE" end of the line amplifier.
3. Place the DB25 female connector on the other end of the 6' adapter cable into an available Com Port of the PC.
4. Place the DB25 female end of the DB25 female to DB9 male adapter into the "To DCE" end of the line amplifier.
5. Place the DB9 female end of the serial cable into the DB9 male end of the DB25 female to DB9 male adapter.
6. Insert the AC power adapter plug into the socket on the line amplifier and plug it into an AC outlet.

[Connect](#)

Serial Cable Construction

Operators should avoid installing serial cables next to high voltage lines and should prevent any foot traffic from occurring over or across the serial cables. The serial cable should not be involved in a situation where damage is probable, and operators who create their own serial cables should not exceed 100'. Serial cables that exceed 100' require the assistance of line amplifiers, and **MultiCam** has tested lengths of serial cable up to 1000' that utilized serial line amplification.

The following is the pin out for the **MultiCam** serial cable. In both cases, Pin #9 is not used.

Computer End, DB9 Female

Pin #	Color
1	(not used)
2	Red
3	Blue
4	White
5	Yellow
6	Black
7	Orange
8	Brown

Machine End, DB9 Female

Pin #	Color
1	(not used)
2	Blue
3	Red
4	Black
5	Yellow
6	White
7	Brown
8	Orange

[Connect](#)

Serial Communication

The **MultiCam** Controller can communicate with a host PC through an RS232 Serial or Com Port. The serial port transmits one bit at a time spread out serially over time. Since only 1 bit rather than 8 is transmitted at a time, only 1 data line exists in the cable for each direction of travel with 2 lines total. A half-duplex allows only 1 data line enabling communication in 1 direction at a time, while the full-duplex allows 2 data lines enabling simultaneous communications in both directions. Thus, a serial port cable is thinner than a parallel port cable.

The port is called an RS232 port because the electrical characteristics follow the EIA Recommended Standard #232, which specifies the operating voltage ($\pm 12V$), relative to a common reference. Other details, such as impedance and driver capacity, are specified in this standard.

The controller's default configuration for options within the standard is 8 data bits, 1 stop bit, no parity, full-duplex, and hardware handshaking. Baud rates supported are 9.6K, 19.2K, 38.4K, and 57.6K bits per second with the standard setting being 19.2K.

The controller operates better when the device transmitting the motion commands obeys DTR hardware handshaking. The sophisticated processing and motion firmware provides continuous motion under most circumstances. In the event that additional processing time is required by the controller and the incoming character buffer fills, the DTR handshaking line will go low to prevent further transmissions of commands from the host PC. When the controller has processed enough of the incoming character buffer, it will then raise DTR and the host PC will resume transmitting.

[Connect](#)

Innovation. Quality. Performance. Best in Industry. Worldwide.

Software

[top](#)

Software	2
AlphaCam Post Processor Installation	2
EnRoute Driver Setup	4
EnRoute View and Preference Settings	8
MultiCam Software	13
Motion Controller Utilities	13
JobNameServer	14
JobEditor	15
JobReporter	15
MotionMechanic	16
ConnectionManager	16
JobConsole	17
JobNameServer Programs	
Digitize	18
HotWatch	18
JobQueue	18
HPGL Command Support	19
HPGL Commands	21
MultiCam M- and G-Code Support	22
Mapping M Code to Output	25
Glossary	27

Software

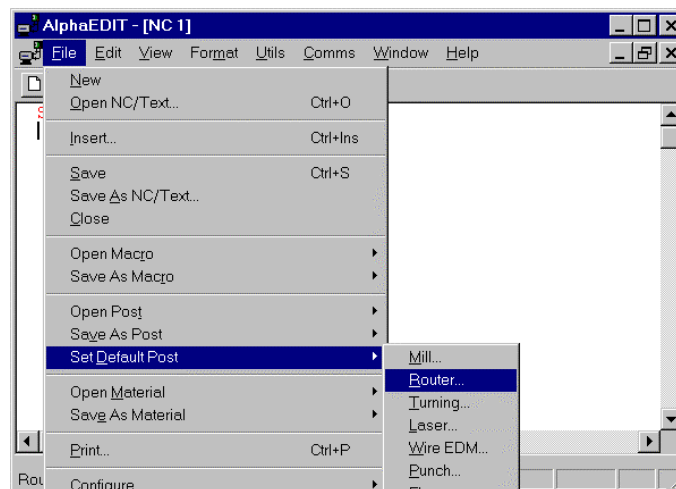
The **MultiCam** Router can use files created from many popular software packages in the industry, such as AlphaCam and EnRoute, as well as files created in G Code and HPGL. The productivity suite enhances communication between the **MultiCam** Router and the host computer. [Software](#)

AlphaCam Post Processor Installation

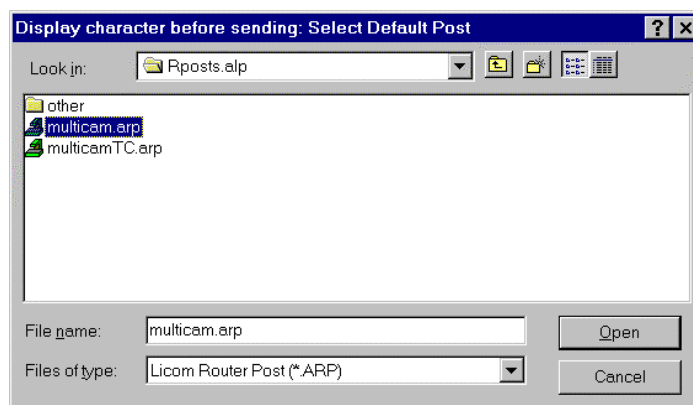
MultiCam offers 2 AlphaCam post processor systems for use depending on the type of post system in the machine. The **MultiCam** ATC-ARP post processor interfaces with both ATC and multiple head systems while the **MultiCam** ARP post processor interfaces with single head systems and outputs a manual tool change command so that multiple tool change files can be run on single head systems.

Multiple Post Systems

1. Copy the appropriate post processor located in Software Drivers\AlphaCam from the **MultiCam** Installation CD into the LICOMDAT\RPOSTS.ALP directory using Windows Explorer.
2. Start the Alpha Edit program from the AlphaCam folder in the Windows Start menu once the post processor is copied.
3. Select Set Default Post – Router from the file menu in Alpha Edit.



4. Select the desired post processor from the list. Click *Open*. This will set the selected post processor as the default.



Single Post Systems

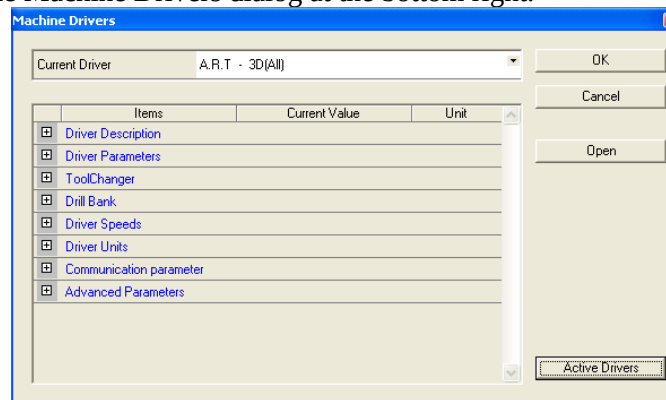
1. Contact **MultiCam** or one of its authorized distributors for a copy of the OnePost file to use with the single post system.
2. Copy the encrypted OnePost file into the LICOMDAT\RPOSTS.ALP directory. This file has no extension.
3. Verify that AlphaCam completes the auto detect of the single post system post processor once the file is copied into the RPOSTS.ALP directory. [Software](#)

EnRoute Driver Setup

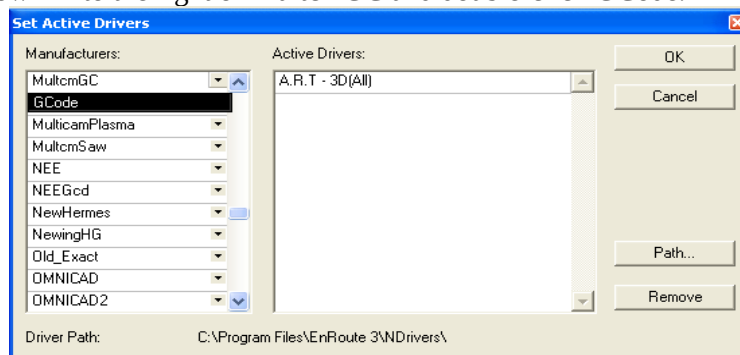
The EnRoute software package is used to assist operators in creating job files for the **MultiCam** Router. Each operator must have both an installation CD to set up the capability and a dongle  to connect with the host PC each time the program is run.

Verifying Computer Compatibility

1. Refer to the EnRoute Recommended System Requirements to make sure the host PC is compatible with the software package.
2. Open EnRoute and select Setup/Machine Setup from the toolbar at the top.
3. Click *Active Drivers* in the Machine Drivers dialog at the bottom right.



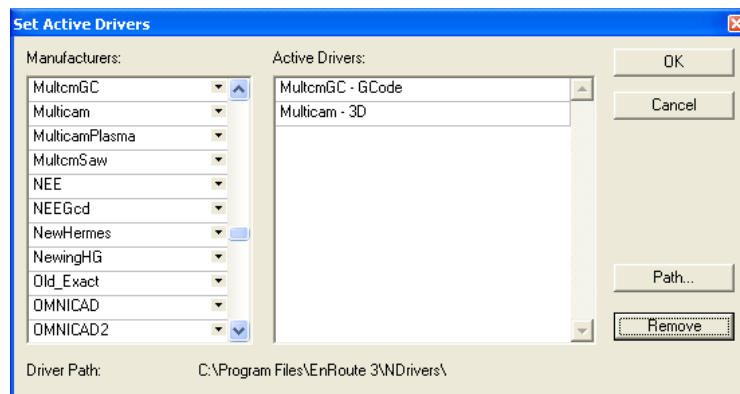
4. Scroll down on the left side to MultcmGC in the Manufacturers Window.
5. Click on the down arrow  to the right of MultcmGC and double-click GCode.



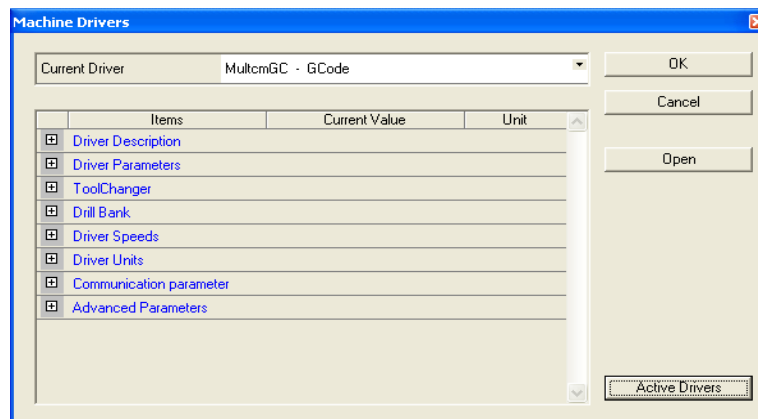
This will add MultcmGC/GCode to the active drivers list on the right.

Selecting the Driver

1. Select the appropriate driver based on the system requirements (e.g., Multicam - 3D Driver [HPGL]).
 - a. **Multicam – 2D** uses the **MultiCam** 2D Driver for machines that **do not** support 3D and will override all cut parameters set at the keypad.
 - b. **MultiCam – 3D** uses the **MultiCam** 3D Driver for machines that **do** support 3D and can be used for cutting bridges, multiple-pass cuts, or 3D machining. This driver will output all cut parameters (e.g., X,Y,Z cut speeds, depth, tool lift, spindle speed, and dwell) and will override any parameters set at the keypad.
 - c. **MultiCam – No Depth** can be used on 2D and 3D **MultiCam** machines and will only output the tool path without outputting any cut parameters (e.g., X,Y,Z cut speeds, depth, tool lift, spindle speed, and dwell) regardless of their settings in EnRoute. All settings must be set on the keypad pendant. This driver will not output 3D.
2. Highlight the default driver (e.g., A.R.T. 3D [All]) and select *Remove*.
3. Select *OK* to accept the appropriate **MultiCam** GC/GCode driver.

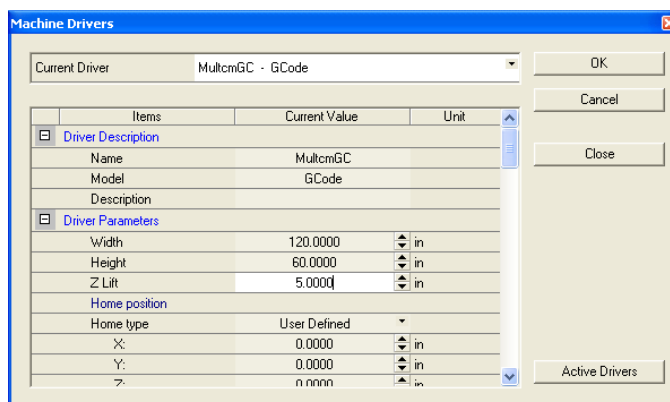


4. Select *OK* to return to the Machine Drivers dialog. The selected driver should be listed at the top.



Adjusting Table Settings

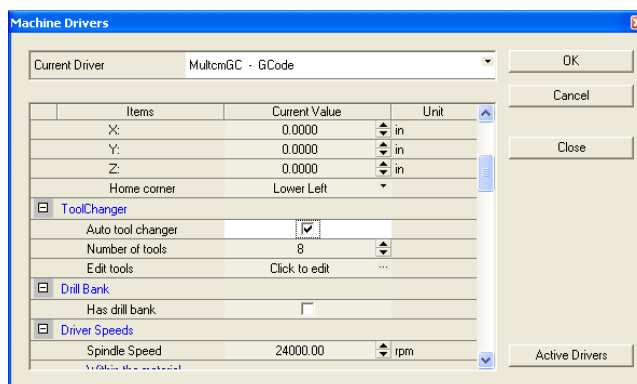
1. Select *Open* to adjust the table settings.
2. Scroll down to Driver Parameters and change the width, height, and Z lift to match the table.
 - a. Width = X-axis (front to back; long side of the table)
 - b. Height = Y-axis (left to right; short side of the table along the gantry)
 - c. Z lift = Z-axis (up and down; Tool Lift Height above the table)



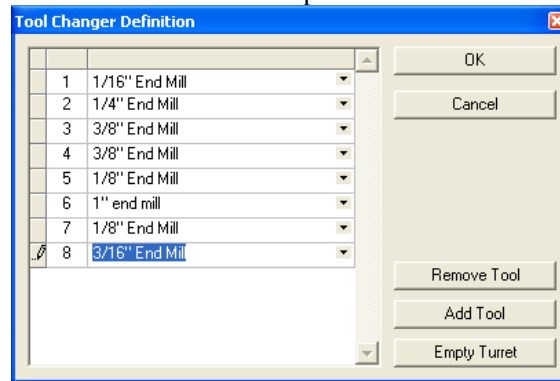
3. Make sure the dimensions are correct before moving on.
 - a. If there is no ATC attached to the Router, the operator may select *OK* and skip ahead to the **Completing EnRoute Setup** section.
 - b. If there is an ATC attached to the Router, the operator should configure the tool changer.

Configuring the Tool Changer

1. Scroll down to Tool Changer and check the box next to Auto Tool Changer.
2. Use the double arrows  next to Number of Tools to correct the file to show 8 for a carousel or 4 for a linear tool changer. The numbers 4 and 8 do **not** have to match the total tools in the carousel.



3. Double-click the *Click to Edit* section located below the selected number of tools and next to Edit tools.
4. Determine how the tools will be set up in the Tool Changer and click on the first down arrow .
5. Select the correct dimensions for each tool from the drop-down menu.



6. Select *OK* when finished.

Completing EnRoute Setup

1. Select *OK* in the Machine Driver Dialog to finish the EnRoute Setup.
2. Make no changes to the Driver Units section as the system is configured with default settings and will not work if these settings are changed. The operator can modify the settings under *Preferences* or the view setup F10.

[Software](#)

EnRoute View and Preference Settings

While **MultiCam** recommends different settings for standard configurations, the operator is able to review the condition of the cutting sequence and the setup before the cutting sequence begins.

View Options

- * Check the screen views to view the associated rulers. The rulers are used to verify size and positioning relative to the location in that Layout View.
- * Scroll up, down, left, or right to pan the viewing area without moving the contours.

Toolbars

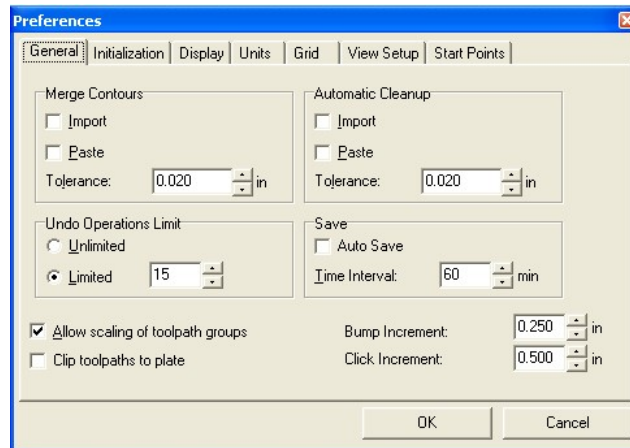
- * Select Setup/Toolbars to choose a toolbar in order to view or hide it. All toolbars will be shown by default.
- * Move a toolbar to the desired location by holding down the left mouse button on the dividing line that separates each bar and dragging it. Once the toolbar is close enough to one of the outer edges of the EnRoute window, the toolbar will snap to it, rotating itself accordingly, and the operator may simply release the mouse button.
- * Drag the toolbar to the layout viewing area and release it to the title bar at the top. The toolbar can then be grabbed by clicking and holding down the left mouse button.
- * Return to the default settings by selecting Setup/Toolbars/Reset to Defaults.

The Preference Settings available in EnRoute 3 (i.e., General, Initialization, Display, Units, Grids, View Setup, Start Points) are used to maximize or speed up the process in which the operator creates a design and applies the tool paths. While the operator has the final say in setting up the program, **MultiCam** recommends the following configurations. More information on a particular setting is available by pressing F1 Help, selecting the Search tab, typing in the tab name (e.g., General tab), and double-clicking the tab name under List Topics.

General

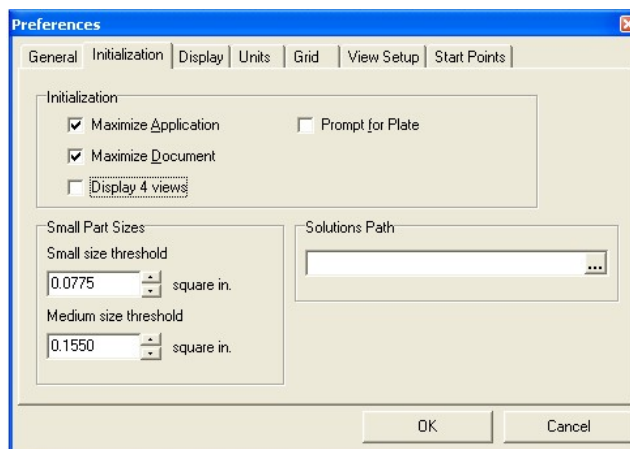
- * Uncheck Merge Contours and Automatic Cleanup options since they may distort or join contours. Both the Merge and Cleanup options, found under Transform, can be completed once the artwork is successfully brought in.
- * Undo Operations Limit due to PC resources and slow operation time. The Limited section should be set around 15 or 20 undos, though the operator would probably be better off starting over if he or she must undo more than 10 times.
- * Undo Allow Scaling of Toolpath Groups to prevent using up resources. Toolpath is included in the size when scaling these groups, and it may be better to delete the tool paths and rescale the contours.

- * Determine where Clip toolpaths to plate is necessary since it allows the operator to output only the tool path groups that are on the plate. This is useful when creating jobs that are too big for one sheet of material but should be kept together. An out-of-bounds message can occur when the operator is running a file as a closed contour, though parts of the file are not cutting, because either the tool path option is checked or the tool path is hanging off the plate.



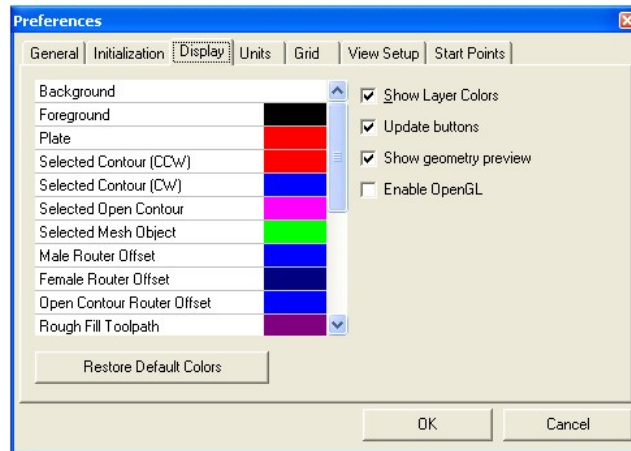
Initialization

- * Select Maximize Application and Document to open the program and any files at maximum window size.
- * Reserve the Display 4 Views option until toolpaths are in place and a particular view should be seen normally. Display 4 Views will show the top, front, side, and perspective view when opening, importing, or starting a new file.
- * Select Prompt for Plate so that the system will prompt at the start of a file to define a plate or material size. A plate does not have to be defined and can be handled when nesting objects or running the 3D simulation. The Clip to Plate option should be unchecked when not defining a plate.
- * Avoid changing the default settings for Small Part Sizes.



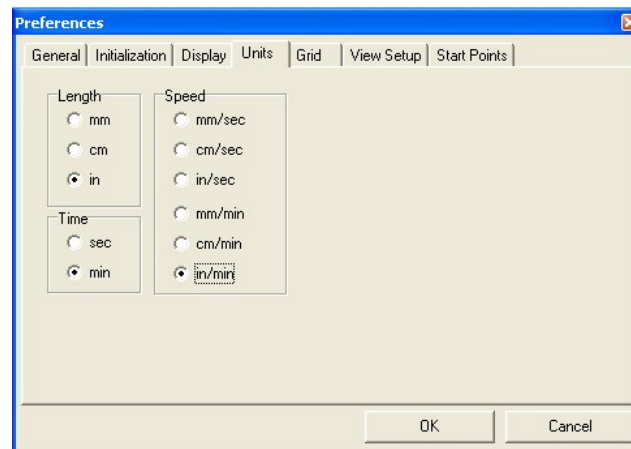
Display

- * Avoid changing the Display Settings from the default setting.
- * Double-click the individual Colors to make any preferential changes for contour appearance during viewing.
- * Leave the 3 options to the far right checked in order to make certain commands user friendly and more functional.



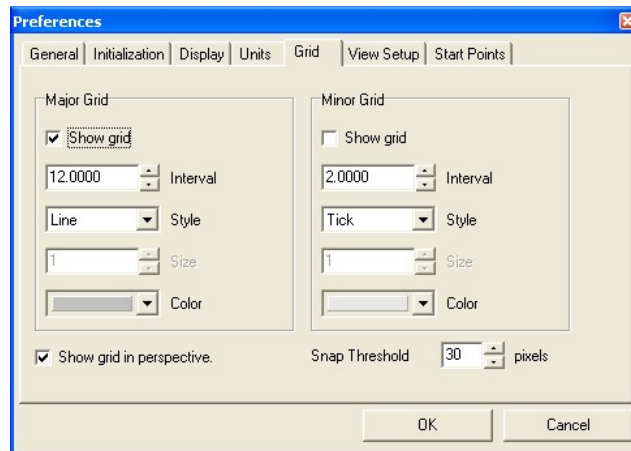
Units

- * Adjust the Length Time and Speed only when working with metric systems as most companies in the US use inches per minute, which is also the basis for most tool catalog formulas.
- * Change unit measurements in this window only. Operators should avoid making any changes in the Machine Setup/Driver configuration.



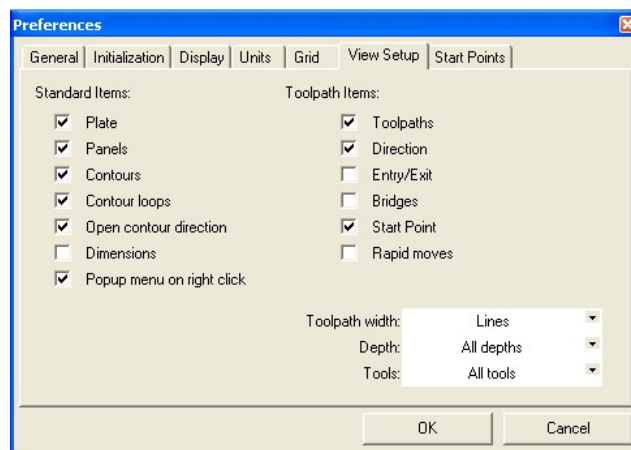
Grid

- * Change the Major and Minor Grids when creating contours in Contour Creation, which can be turned off when not in use.
- * Review the parameters of the grid. The Interval describes the size of the grid while Size describes the pixel size of the Style.



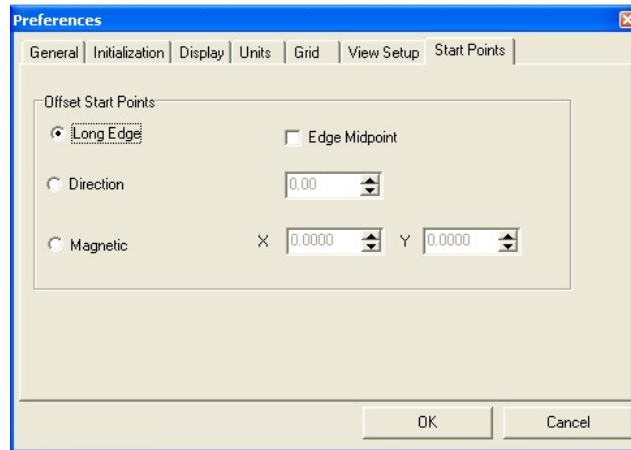
View Setup

- * Adjust View Setup settings to show or hide the listed items when creating contours and tool paths.
- * Avoid changing the options in the white window until all paths and options for them have been created. At that point, these settings can be changed to see only certain paths with these 3 options.



Start Points

- * Avoid changing the Start Points settings from the default setting since these offset points can be adjusted manually during the cutting sequence.
- * Review the Long Edge configuration as this places the start point on the longest edge of the geometry.
- * Select the Edge Midpoint only if the start point should be relocated to the middle of the longest edge.



[Software](#)

MultiCam Software

The **MultiCam** software installation CD contains the **MultiCam** Motion Controller software, which provides the interface between the computer and the **MultiCam** Router. The installation CD also contains machine schematics, software drivers, and a routing guide that contains tips on different cutting tools.

Before installing **MultiCam** software, customers should make sure there are no other applications open on the computer. Internet Explorer v5.0 or newer is required for this installation.

A. Insert the **MultiCam** Installation CD into the CD-ROM drive. This will normally cause the computer to read the CD and then direct the customer through the commands for installation.

B. Follow these steps if the installation does not start automatically:

1. Select the Windows Start button at the bottom left of the screen.
2. Select *Run*.
3. Click *Browse* on the Run Dialog box.
4. Select the Multicam Installation Folder from the directory.
5. Select Setup.Exe.
6. Click *Open*.
7. Click *OK* on the Run dialog box.
8. Follow the setup menus for a complete installation of the **MultiCam** software.

[Software](#)

Motion Controller Utilities

The **MultiCam** software was developed to provide the operator more options and to interface with the newer Windows software. In addition to supporting the DNC function, the **MultiCam** software also has options for digitizing, as well as JobReporter, bar code scanning, and tool diameter compensation. These tools are all written in 32-bit code and run directly from Windows 2000 or higher. Operators can access additional information on these features in C:/Program Files/Machine Tools Suite4.

Suite4



[JobNameServer](#)



[MotionMechanic](#)



[JobEditor](#)



[ConnectionManager](#)



[JobReporter](#)



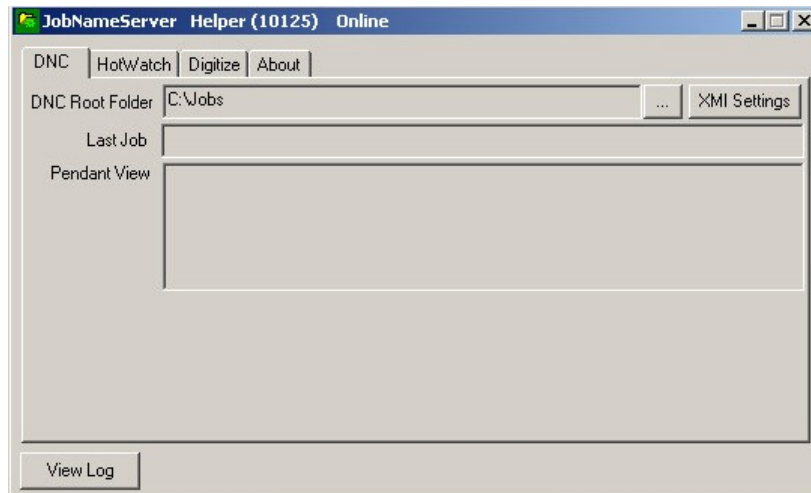
[JobConsole](#)

[Software](#)

JobNameServer



The JobNameServer program connects the operator with folders on the network or host PC by finding jobs quickly and executing them from the keypad. Other features of JobNameServer can help automate the production process. Operators can click on the tab and start a program or learn about the system.



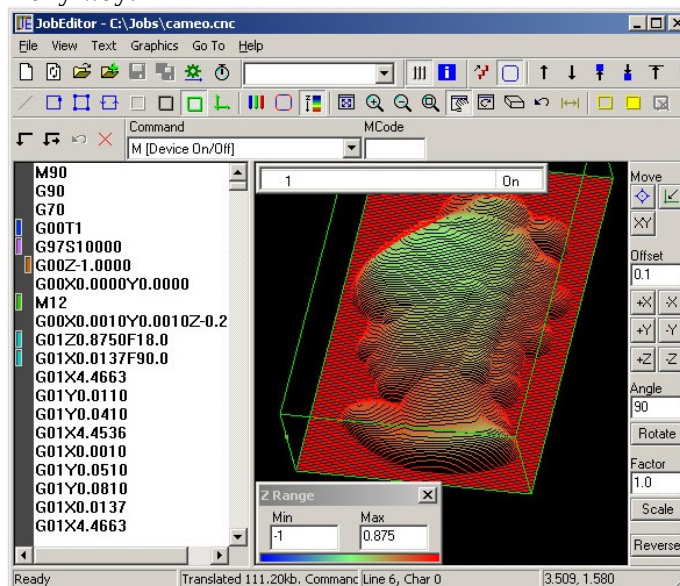
- a. **DNC** allows the operator to select and execute jobs from the network or host PC using either a keypad or touch screen interface.
- b. **HotWatch** helps automate the production process by monitoring a folder on the network and automatically executing jobs that are placed into that folder.
- c. **Digitize** helps operators reproduce 2D and even 3D objects. Not all machines support Digitize, and operators can contact **MultiCam** for additional information. A machine supporting Digitize uses special sensors to scan the object and create a job file based on the object. Once the object has been digitized, the job file can be executed as many times as necessary
- d. **About** shows the versions of JobNameServer and the associated files used by JobNameServer.

[Software](#)

JobEditor



The Job Editor application allows operators to preview and edit jobs through resizing, repositioning, or modifying current job files in any way.

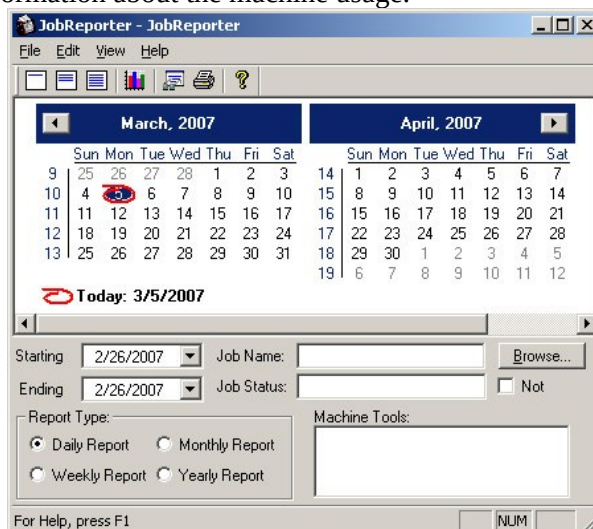


[Software](#)

JobReporter



The JobReporter generates reports on machine use. The report results can be filtered and modified to obtain general or specific information about the machine usage.

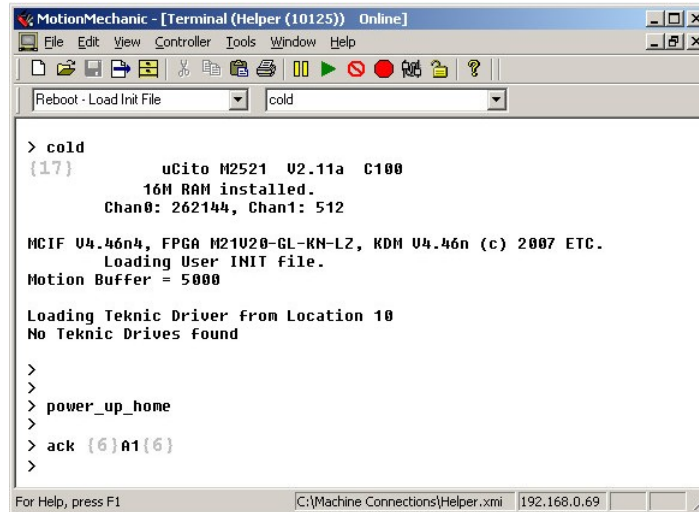


[Software](#)

MotionMechanic



The MotionMechanic application provides the operator with advanced technical control over the use and setup of the motion controller from a host PC.

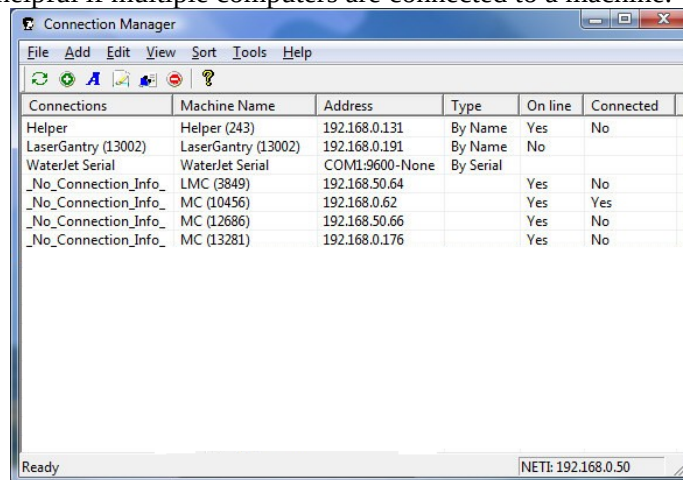


[Software](#)

ConnectionManager



The ConnectionManager helps in creating and configuring the connection between a host PC and the machine tool and has built-in testing features for diagnosing communications issues and sending email correspondence for assistance. Operators can also establish one particular computer to be in charge of all communications, which is helpful if multiple computers are connected to a machine.

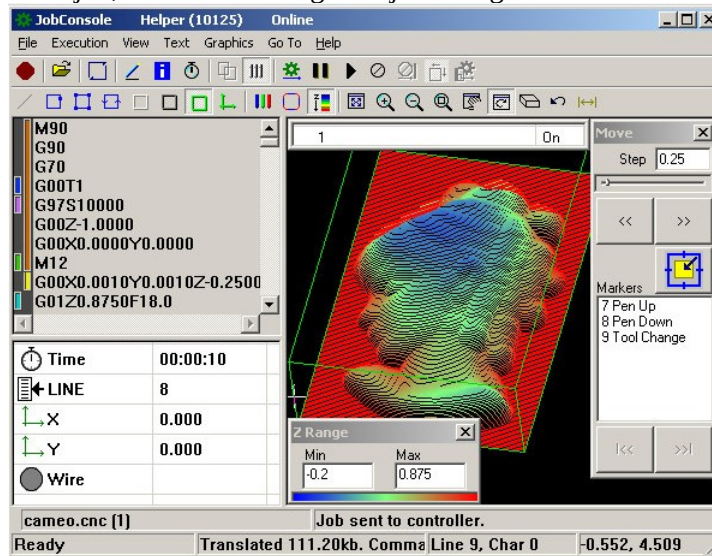


[Software](#)

JobConsole



The JobConsole program acts as the main graphical interface between a machine tool and the operator, allowing the operator to preview and execute jobs as well as monitor the job progress, pause and resume the job, move to a specific point in the job, and move through the job a single cut at a time.



[Software](#)

JobNameServer *Digitize*

The Digitize component of JobNameServer allows the operator to Jog the machine around and trace a pattern to record movements that will be played back later, thus simplifying new job file creation, with several Digitize Commands.

Using the Digitize functions allows the operator to create a G-Code (CNC) job file by tracing a pattern. Once the pattern is traced, the job file can be saved and reviewed. Once the file is saved, the job is easily executed using DNC. Digitize is only operable with a **MultiCam** Router equipped with the Edge Scan initialization file load in the controller. [Software](#)

JobNameServer *HotWatch*

The HotWatch component of JobNameServer allows job files to be automatically transferred to the **MultiCam** Router by monitoring a specified directory on the computer. Whenever a file with a known extension appears in this Inbox directory, HotWatch will transmit it to the router and then move the file from the Inbox directory to the Outbox directory. These directories do not have to be located on the local hard drive but can exist as shared folders on a local or wide area network. HotWatch can be configured to check for new files at specified intervals to help minimize network traffic.

The full path of the Inbox and Outbox directories is displayed in the HotWatch applet, and these directories can be changed at any time by clicking on the Browse button to select another path. The HotWatch Status informs the operator of all activities that HotWatch is performing.

HotWatch does not automatically monitor the Inbox directory or transmit job files. Operators must manually set up HotWatch and activate this function. [Software](#)

JobNameServer *JobQueue*

The JobQueue component of JobNameServer allows jobs for production runs to be created and set up. The job file must be created and started before JobQueue will execute, and the operator can set up JobQueue in the JobQueue window. Once these settings are made and JobQueue is started, the operator will be prompted to start each job after seeing the repetition number of the job being executed. This allows the operator to move to another home position or unload and load material before the cutting sequence begins. [Software](#)

HPGL Command Support

The **MultiCam** control system internally translates HPGL. Feedrate, Depth, and Tool Lift can be set by the file or by the machine's keypad interface. If the Feedrate, Depth, and Tool Lift are set at the machine prior to the file being sent, the file will override all those values set at the machine. If no value is sent in the file, then the machine will use the value programmed at the keypad. The control system ignores the HPGL VS command.

At the beginning of the file, the controller **must** be sent the IN command to begin processing plot commands. If the letter I is **not** the first character sent, then a linefeed or semi-colon must be sent to re-initiate the sequence. A semi-colon must follow the IN command. If any other character is sent during this sequence, then the complete sequence must be restarted. This command sequence must be sent again if an SP or SP0 command is sent. The recommended initialization string is “;IN;” and the recommended termination string is “SP0;”. All files should be written in absolute mode. Some of the early routing systems do not support relative arc commands.

HPGL Resolution: 1016 inch =40 Metric

The following pages contain all of the supported commands. Bracketed information is optional, while braced information is exclusive. A semi-colon or a linefeed **must** terminate all commands, and spaces **cannot** be substituted for required commas in the syntax of a command. All Z-axis arguments are only valid if the 3D mode is set. All commands are integers unless otherwise specified.

Help documents on the HPGL and CNC systems are located in the **MultiCam** folder, and operators can access this folder through either Start/Programs/Multicam/Help/Translated PLT Help or Start/Programs/Multicam/Help/Translated CNC Help.

Example Files

Multi Tool File

```
;IN;PU;ZZ0;  
PA0,0;  
TCTool 1;  
ZD127;ZU400;  
PA1016,1016;PD1016,1016;PD11176,1016;  
PD11176,11176;PD1016,11176;PD1016,1016;  
PU;TCTool 2;  
ZD127;ZU400;  
PA2032,2032;PD2032,2032;PD10160,2032;  
PD10160,10160;PD2032,10160;PD2032,2032;  
PU;TCTool 3;  
ZD127;ZU400;  
PA3048,3048;PD3048,3048;PD9144,3048;  
PD9144,9144;PD3048,9144;PD3048,3048;  
PU;PA0,0;  
SP0;
```

Automatic Tool Changer File (or Multi Head)

```
;IN;PU;ZZ0;  
PA0,0;  
SP3;  
ZD127;ZU400;  
PA1016,1016;PD1016,1016;PD11176,1016;  
PD11176,11176;PD1016,11176;PD1016,1016;  
PU;SP1;  
ZD127;ZU400;  
PA2032,2032;PD2032,2032;PD10160,2032;  
PD10160,10160;PD2032,10160;PD2032,2032;  
PU;SP2;  
ZD127;ZU400;  
PA3048,3048;PD3048,3048;PD9144,3048;  
PD9144,9144;PD3048,9144;PD3048,3048;  
PU;PA0,0;  
SP0;
```

Arc File

;IN;ZZ0;SP1;
PA1360,2450;ZD0.500000;PD;
AA1270,2540,360.000000;PU;
PA2010,3048;PD;PA1292,3766;PU;
PA2630,3720;PD;AA2540,3810,360.000000;
PU;PA3290,4263;PD;PU;PA4154,3720;
PD;AA4064,3810,360.000000;
PU;PA4902,4263;PD;PU;PA5678,3720;
PD;AA5588,3810,360.000000;PU;
PA4064,2921;PD;PA2540,2921;
AA2540,2540,180.000000;PA5588,2159;
AA5588,2540,180.000000;PA4064,2921;
PU;PA4154,1180;PD;AA4064,1270,360.000000;
PU;PA5678,1180;PD;AA5588,1270,360.000000;
PU;PA6948,2450;PD;AA6858,2540,360.000000;
PU;PA2630,1180;PD;AA2540,1270,360.000000;
PU;PA7747,2540;PD;AA5588,2540,-90.000000;
PA2540,381;AA2540,2540,-
180.000000;PA5588,4699;
AA5588,2540,-90.000000;PU;PA8128,0;SP0;

Sample 3D file

IN;ZZ1;PU;SP1;
PA5000,5000,-200;
PD5000,5000,500;
PA5000,15000,1000;
AA10000,15000,-180;
PA15000,5000,750;
AA10000,5000,-180;
PA5000,5000,-200;
PU;SP0;

Multi Pass File without Lifting the Z

;IN;PU;
PA0,0;
SP1;
ZD100;
PA1016,1016;
PD1016,1016;PD11176,1016;
PD11176,11176;PD1016,11176;PD1016,1016;
ZD200;
PD1016,1016;PD11176,1016;
PD11176,11176;PD1016,11176;PD1016,1016;
ZD300;
PD1016,1016;PD11176,1016;
PD11176,11176;PD1016,11176;PD1016,1016;
ZD400;
PD1016,1016;PD11176,1016;
PD11176,11176;PD1016,11176;PD1016,1016;
PU;SP0;

[Software](#)

HPGL Commands

Initialization

IN instructs the controller to begin processing the HPGL plot file. Without this, the commands in the file are received but never executed. If multiple IN commands are found during the execution of the file, the controller performs a Pause/Cancel operation. All motion from the previous job that has yet to be executed is lost, and the new information is executed. When the IN command is executed, the HPGL 3D mode is cleared. The 3D mode can be reset using the ZZ command. A PU command will be executed before the system executes the first HPGL move command.

ZZ (0,1) establishes that a value of 0 sets 2D mode and a value of 1 sets 3D mode. In 2D mode, all commands must have 2 axes, while all commands must have 3 axes in 3D mode.

Motion

PA[X,Y][,Z] moves to an absolute HPGL position and sets absolute mode for future PU and PD commands. If no arguments follow the command, only absolute mode is set.

PD[X,Y][,Z] executes a Pen Down then moves to the requested position, if one is specified. This position is dependent on whether absolute or relative mode is set. The Feedrate Override knob is turned ON.

PU[X,Y][,Z] executes a Pen Up then moves to the requested position, if one is specified. This position is dependent on whether absolute or relative mode is set. The Feedrate Override knob is turned OFF.

AA X Ctr,Y Ctr,Angle is a floating point requiring a non-integer value that draws an arc with its center at X,Y. A positive angle creates a counter-clockwise arc.

ZA Z moves the Z-axis to the absolute position specified. This feature is for both 2D and 3D modes.

Speed

SF[XY-rate],[Z-rate], floating points, sets the feedrate for the machine tool during Pen Down. All rates are in mm/sec. The Z rate applies to the plunging feed rate of the Z-axis. In 3D mode, the Z-axis rate is not used.

Other

SP[0, 1, 2, ..., 8...] selects a new tool for use. If zero or no pen number is given, the controller performs an end of file command (EOF). Once an EOF is performed, no motion is executed until a new IN command is received. Pens 1—8 are spindles, and pens 11—12 are drills.

ZD Z, floating point, sets the distance from the surface for the Z-axis to move down on a PD command (Depth). Changing the ZD value will change the depth on all following PD commands. The value set with this command is **not** used when running in 3D mode.

ZU Z, floating point, sets the distance from the surface for the Z-axis to move up on a PU command (Tool Lift). Changing the ZU value will change the lift distance on all following PU commands. The value set with this command is **not** used when running in 3D mode.

TC [text] issues a tool change command. The text following the TC command will appear on the keypad display at the appropriate time in the tool change sequence. The operator will have the ability to surface the new tool, change speeds, depths, and the lift. If this information is going to be set by the file, the appropriate commands must be sent after the tool change command. [Software](#)

MultiCam M- and G-Code Support

The following table lists the supported G and M Codes for the **MultiCam** Controller. Parameters within brackets are optional. The fields represented by “d.d” may be any decimal number, and fields represented by “d” may be any positive integer. All lines of machine code must start with either an M Code or a G Code.

G00 [Xd.d] [Yd.d] [Zd.d] [Fd.d] [Td] [Ctext string] - High speed move (slew)
G01 [Xd.d] [Yd.d] [Zd.d] [Fd.d] - Linear move (machine)
G02 [Xd.d] [Yd.d] [Zd.d] [Id.d] [Jd.d] [Kd.d] [Fd.d] - CW 2D circular move
G03 [Xd.d] [Yd.d] [Zd.d] [Id.d] [Jd.d] [Kd.d] [Fd.d] - CCW 2D circular move
G04 Fd.d - Dwell (seconds)
G17 - Specify XY plane for helical
G18 - Specify ZX plane for helical
G19 - Specify YZ plane for helical
G37 - Find home
G62 - Clear soft home
G70 - English programming (inches)
G71 - Metric programming (mm)
G72 [Xd.d] [Yd.d] [Zd.d] [Id.d] [Jd.d] [Kd.d] [Fd.d] - CW 3D circular move
G73 [Xd.d] [Yd.d] [Zd.d] [Id.d] [Jd.d] [Kd.d] [Fd.d] - CCW 3D circular move
G74 - Incremental mode for G02/03 arcs
G75 - (G90/G91) mode for G02/03 arcs
G83 Rd.d Zd.d Dd.d [Fd.d] - Peck drill
G90 - Absolute coordinate mode
G91 - Incremental coordinate mode
G92 [Xd.d] [Yd.d] [Zd.d] - Set soft home
G97 Sd - Set spindle speed (rpm)
M00 - Program pause
M01 - Optional pause
M02 - Program end
M11 - 2D device on (like HPGL PD)
M21 - 2D device off (like HPGL PU)
M12 - 3D device on
M22 - 3D device off
M30 - AUX 4 on not affected by the Pause button
M31 - AUX 4off not affected by the Pause button
M90 - Program start
M99 - Exit CNC interpreter

The following table lists the letters used to denote various arguments in ETC CNC version 1.0.

C - Tool change operator message (used in G00)

D - Peck drill delta (used in G83)

F - Feed rate (used in G00, G01, G02, G03, G72, G73, G83)

F - Dwell (used in G04)

G - Preparatory function

I - Circular interpolation value in X dimension (used in G02, G03, G72, G73)

J - Circular interpolation value in Y dimension (used in G02, G03, G72, G73)

K - Circular interpolation value in Z dimension (used in G02, G03, G72, G73)

M - Miscellaneous function (control function)

N - Sequence number

R - Beginning Z motion dimension (used in G83)

S - Spindle rpm (used in G97)

T - Tool change (used in G00)

X - X motion dimension

Y - Y motion dimension

Z - Z motion dimension

The following is a 5" square with a rapid level of 0.5" above the material, feed down at 100 ipm, cut feed at 200 ipm, rapid down to 0.1" above the material, and a cut depth of 0.25".

M90

G90

G70

G75

G00 T1

G00 Z-0.5

G00 X0. Y0.

M12

G00 Z-0.1

G01 Z0.25 F1.667

G01 X5. F3.333

G01 Y5.

G01 X0.

G01 Y0.

G00 Z-0.5

M22

G00 X0. Y0.

M02

Innovation. Quality. Performance. Best in Industry. Worldwide.

The following is a 5" circle clockwise, with a center at 2.5,2.5, rapid level of 0.5", feed down at 60 ipm, cut feed at 120 ipm, rapid down to 0.1", and a cut depth of 0.4".

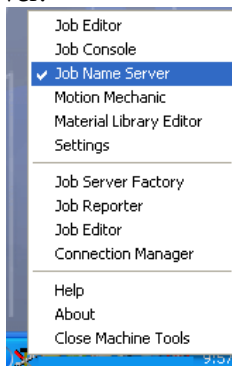
M90
G90
G70
G75
G00 C1/2"bit
G97 S18000
G00 Z-0.5
G00 X2.5 Y2.5
M12
G00 Z-0.1
G01 Z0.4 F1.
G02 I2.5 J2.5 F2.
G00 Z-0.5
M22
G00 X0. Y0.
M02

[Software](#)

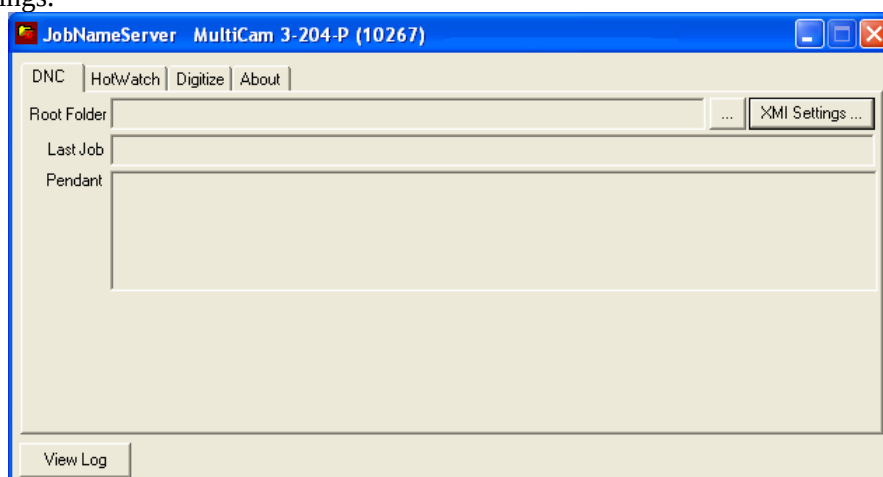
Mapping M Code to Output

Operators can add M Codes to an output on their existing machines, assign the code to a particular device, and set the active status by completing the following steps:

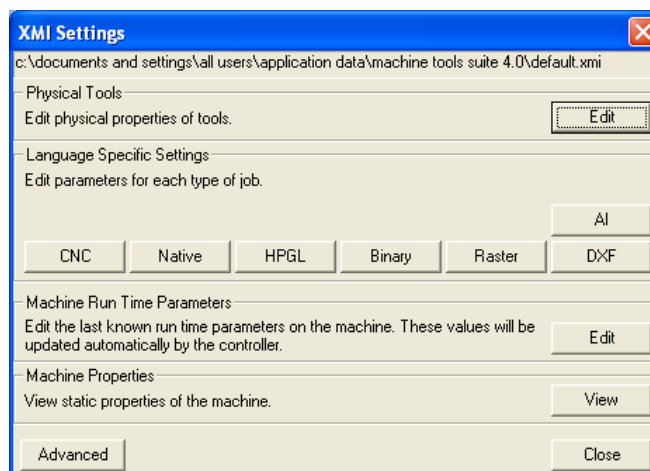
1. Open MachineTools and select JobNameServer.



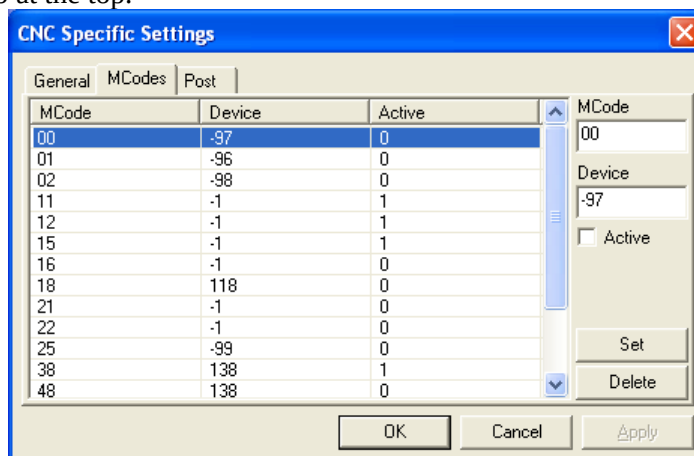
2. Select XMI Settings.



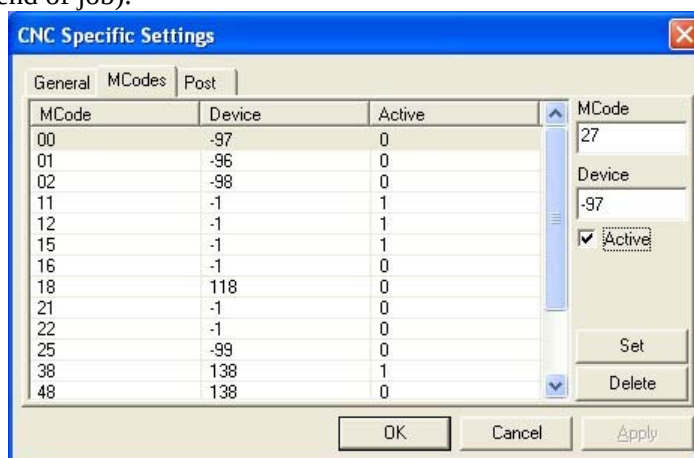
3. Select CNC.



4. Click on the MCodes tab at the top.



5. Enter the M Code (e.g., 27), correct the device as needed, set the status to Active by clicking in the check box, and select **Set**. **Active** indicates a motion command (e.g., start spindle) while **Inactive** indicates a non-motion command (e.g., end of job).



Any M Code, including M12, can be mapped per the operator's preference; however, JobConsole treats the M12/M22 coding specifically for the start/end of a contour.

6. Download and store an MCode_Device_Map.uc file to location 3. For assistance on any of these steps, please contact **MultiCam** Tech Support.

Operators may also modify or delete M Codes from this same dialog.

Modify - Highlight an existing M Code, make changes (e.g., Device -1 instead of -97), and select **Set**.

Delete - Highlight an existing M Code and select **Delete**.

[Software](#)

Glossary

The following words are most commonly used in reference to the router and associated activities.

Acrylic - A type of plastic, or a common material used in sign making.

Aggregate - A type of tool that is not vertical and does not spin in line with the spindle axis. An aggregate tool may also rely on the C-axis for operation (e.g., saw).

Axis - A direction in a coordinate system. **MultiCam** Routers have 3 axes: X, Y, and Z.

Axis Motor - A motor that causes motion in a particular axis.

Bar Code Scanner - A device that allows operators to retrieve information encoded in a bar code label so that specific notes and directions are kept in line with the correct job file.

Bearing Car - The bearing on which an axis moves along a rail, also known as a bearing truck. There are 2 bearing cars for each rail.

Bearing Rail - The rail on which the bearing cars ride. There are 2 rails for each axis.

Bite Size - The depth of each pass, used in multipass.

Chuck - A clamping device for holding a drill bit.

Climb Cut - A direction the router moves along a cut to produce a climbing motion. For a closed contour, the climb cut is clockwise on the outside of the cut and counter-clockwise on the inside of the cut.

Collet Wrench - A wrench used to loosen or tighten the cover nut on a spindle or tool holder.

Compression Cutter - A spiral up/down cutter. This cutter is used for high feeds in wood and laminates and is configured with a spiral up at the bottom and spiral down at the top.

Conical Cutter - A V-shaped cutter used in relief carving, engraving, and chamfering.

Conventional Cut - The opposite cut of a climb cut, or closed counter-clockwise on the outside of the cut and clockwise on the inside of the cut.

Cross-over Cable - The type of Ethernet connection that is directly connected to the computer.

Cut Conditions - The most efficient condition for standard cutting or engraving. Cut conditions change the motion parameters of the routing system.

Cut Depth - A Hot Key function that sets the depth of cut. Cut Depth refers to the Z-axis distance the end of the cutter will go below the surface of the material during a cut.

Cut Speed - A Hot Key function that sets the speed of the cut.

Dieline - The path taken by the cutting head to complete the cutting sequence. Dielines are most often used in software programs like Adobe Illustrator.

DNC - The system that allows the controller to access sub-directories on the hard drive of a host PC and is used to transfer files from the PC to the controller.

DNC Log - A listing of all activity completed by the DNC and kept by the DNC.

Drill Attachment - An attachment that allows drilling in addition to the routing spindle.

Dry Run - A controller function that will execute a file with no Z-axis movement. Dry Run is used to show the operator where the tool will go during the cutting sequence.

Emergency Stop - The red mushroom button located on the keypad, as well as on the side of the router, used to remove power from the machine excluding the controller board and limit switches.

External Halt - The keypad display that appears when power is applied to the system with the Emergency Stop button pushed in on the keypad pendant.

Feedrate Override - A Hot Key used to adjust the cutting feedrate of the system while a file is being executed.

Fiducial - A reference mark, or dot, used in MultiVision that allows the CNC machine to recognize the orientation of the material and adjust the cut file accordingly.

Firmware - The operating system of the controller that should only be loaded when instructed by **MultiCam**. Firmware contains all of the low-level commands and is the first level of controller software.

Flash RAM File - A file containing all of the machine parameters of a particular router. Settings such as table size and resolution are located in the Flash RAM File, which is the third level of controller software.

Gantry - The mechanical part of the router, also known as the bridge, which moves the carriage assembly from the front to the rear of the table.

G Code - The machine code language used by the controller to execute motion commands.

Gearbox - The drive system on Pro series that uses planetary gear reduction to produce rotation of the pinion.

Glide Pad - The attachment for the pressure foot that glides along the material to keep the material in place.

Hard Home Position - The home position determined by the location of the targets and limit switches, typically the front right side of the table.

Head - A reference to a particular cutting tool (i.e., spindle).

Head Mode - A designation for multiple head machines, either All or Auto.

Hot Keys - The one-touch keys on the keypad that perform controller functions.

HPGL - The machine code language that the system executes as a file.

Init File - The second level controller software that contains high-level commands.

JobNameServer - The motion controller utility program used to communicate between the controller and the host PC in Suite4.

Kerf - The spacing between parts. The kerf must be larger than the tool diameter.

Keypad - The part of the pendant with the grid of Hot Keys.

Light Curtain - A boundary around the machine created by parallel, infrared light beams that stops the cutting sequence once an object or person crosses the beam.

Linear Tool Changer - The tool changing system with the tool holders lined up along an axis. This is the most common tool changer for the SF series.

Machine Parameters Menu - A tool of the **MultiCam** Productivity Software Suite that allows the flash RAM file to be modified.

Menu System - The commands that are not assigned to Hot Keys.

Misting Nozzle - The tip of the flex hose on the misting system.

Misting Unit - The unit that rides on the back of the carriage assembly and provides misting lubrication for cutting metal, available in automatic and manual versions.

Multiple Passes - A cutting movement through material that repeats the contour at different depths with deeper cuts each time based on the bite size.

Multiple Spindles - A router configuration with more than 1 spindle or routing head.

Node - A computer or device that is connected to a Class C or Private network, which is common in most businesses.

Nose Cone Brush - The brush attachment for the pressure foot. The nose cone brush is used when the material is easily scratched by the glide pad.

O Flute - A type of cutter that has a half-moon geometry from the end view.

Operator Station - The stand-alone board separate from the machine and keypad that allows the operator to turn on auxiliary systems or access dual systems if applicable.

Park - The function used in all 3 axes to place the carriage assembly at an established or farthest point away from the material for loading and unloading.

Patch Cable - The type of Ethernet connection that is connected to the computer through a networking component such as a hub, switch, or router.

Pause - The yellow button on the keypad that puts the controller into Pause mode where action is momentarily stopped and can be restarted at the keypad.

Pendant - The control module consisting of the keypad and mechanical inputs.

Pressure Foot - An attachment to the spindle that keeps the material in place and removes chips and other types of debris from the cut material.

Programmable Soft Home - The 9 possible home positions that the operator can store to be recalled at a later time.

Proximity Restart - The controller function that allows the operator to restart a file along any cut move after an interruption such as power outage or tool breakage.

Rack and Pinion - A system of transferring movement from the rotation of the motors to the linear movement of the system by way of gears.

Rotary Tool Changer - The tool changing system with tool holders held on a rotating turret. The rotary tool changer is used on the PRO series.

Sacrifice Material - The material that is placed between the tabletop and the material being cut, also known as spoil board. The sacrifice material is used to protect the table surface and is considered a consumable item.

Self Test - A controller function that allows the system to test itself. The self test is accessed from the menu items and installing the self-test plug on the controller.

Serial Cable - The RS232 cable used to transfer data from the host PC to the controller.

Servo Drive - The servo amplifier used to move the servo motors.

Servo Motor - The axis motor on a servo system.

Side Cover - The covers on each end of the gantry protecting the X-axis motor and limit switches.

Single Flute - A type of cutter that only has 1 cutting edge.

Slew - A non-cutting movement of the system above the material at maximum speed.

Soft Home Position - Any set home position used as the origin for executing programs.

Spindle - The rotating motor that holds the cutting tool.

Spindle Plate - The plate that mounts the spindle to the Z-carriage plate.

Spindle Speed - The speed the spindle rotates, shown in RPMs.

Spiral Downcut - A cutter with flutes that spiral around the shaft. A spiral downcut pushes down the chips that are cut.

Spiral Upcut - A cutter with flutes that spiral around the shaft. A spiral upcut pushes up the chips that are cut.

Spoil Board - The material that is placed between the tabletop and the material being cut, also known as sacrifice material. The spoil board is used to protect the table surface and is considered a consumable item.

Stepper Drive - An amplifier that controls the action of the stepper motors.

Stepper Motor - An axis motor that uses stepper technology.

Straight Flute - A type of cutter with flutes arranged parallel to the center of the cutter shaft.

Surface - The topside of the material used as the Z-axis reference point.

Tool Lift - The Z-axis distance between the bottom of the router and the material. The Tool Lift is also the location of the router during a slew move.

Tool Offset - The distance between the center of the router and the cutting edge. The Tool Offset is also the distance between the desired geometry and the center of the tool path, usually the radius of the router.

Tool Path - The path that the center of the router takes during a cut.

Transmission - The drive system on SF series using a 4:1 belt drive pulley system to produce rotation of the pinion.

Two Flute - A type of cutter that has 2 cutting edges.

µCito - The native language of the controller (pronounced micro-see-tow).

Vector - A line segment showing a particular direction or path.

Vinyl Knife - An attachment that is used to cut vinyl.

Warmup - The process a system goes through to prepare for regular operations (e.g., spindle warmup before any routing begins).

Wood - A common routing material.

X-axis - The axis that refers to the length of the table, front to back.

X Motor Plate - The plate attached to the side of the gantry in which the transmission or gearbox assemblies are bolted.

Y-axis - The axis that refers to the width of the table, left to right.

Y Carriage Plate - The vertical plate behind the Z-axis carriage plate where Z-axis bearing rails are mounted.

Y Motor Plate - The horizontal plate attached to the Y-carriage plate where the Y-axis transmission or gearbox assemblies are bolted.

Z-axis - The axis that refers to the accessible range of the area above the table.

Z Carriage Plate - The vertical plate on the vertical moving part of the carriage assembly where the cutting tool plate is mounted.

Z Motor Plate - The horizontal plate on top of the Y-carriage plate where the Z-axis motor is mounted.

Z Speed - The vertical speed of the spindle assembly or the plunging speed for spindle machines.

[Software](#)